VULCAN

SIMULATION SOFTWARE
FOR CASTING PROCESS OPTIMIZATION

Version 10.0

Tutorials

© Copyright 2007
Quantech ATZ S.A.
Barcelona, Spain

This tutorial manual may not be reproduced in whole or in part, or processed by computer or transmitted in any form or by any other means, whether electronic, by photocopy, by recording or any other method, without the prior consent in writing of the owners of the Copyright.

Quantech ATZ S.A.
Edificio NEXUS
Gran Capitán, 2-4
08034 Barcelona
SPAIN

Phone: +34·932 047 083
Fax: +34·932 047 256
Email: info@quantech.es
http://www.quantech.es

Foreword

This book consists of 8 guided tutorials that will serve as a starting point for using Vulcan. The time that will take to complete these tutorials will depend upon the user's skills in the related topics that involve casting processes' computer simulation.

The tutorials are thought to be completed as a self-training guide for a Vulcan new user, although technical assistance by Quantech ATZ might be necessary.

The tutorials may require some computer files as a starting point (geometries, calculations, etc.), which will be provided by Quantech ATZ.

In order to successfully complete these tutorials, we recommend to also having a copy of the Vulcan User's Manual as a reference guide.

[This page is intentionally left blank]

INDEX

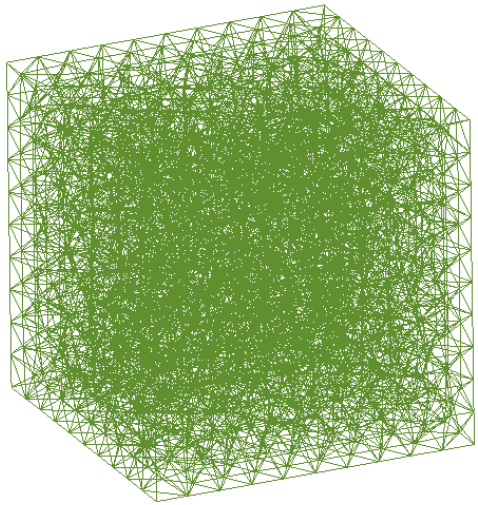
Tutorial	Subject
A	Initiation: basic tools for geometry
B	Tutorial: Implementing a mechanical part
C	Tutorial: Implementing a cooling pipe
D	Geometry preparation in other CAD systems
E	Geometry correction: tools for correcting geometries once imported into Vulcan environment
F	Tutorial of a complete casting process: Gravity casting
G	Tutorial of a complete casting process: High Pressure die casting
H	Tutorial of a complete casting process: Low Pressure die casting

***NOTE**: For beginning level users, we recommend to complete the tutorials in the given order.*

[This page is intentionally left blank]

A. INITIATION TO VULCAN

With this example, the user is introduced to the basic tools for the creation of geometric entities and mesh generation.



[This page is intentionally left blank]

FIRST STEPS

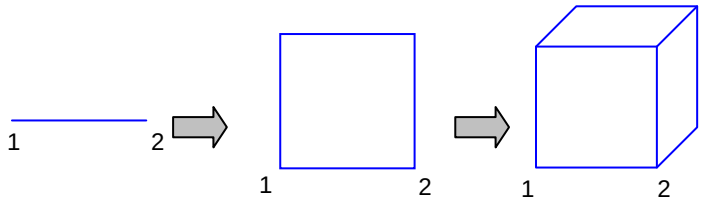
Before presenting all the possibilities that **Vulcan** offers, we will present a simple example that will introduce and familiarize the user with the **Vulcan** program.

The example will develop a finite element problem in one of its principal phases, the preprocess, and will include the consequent data and parameter description of the problem. This example introduces creation, manipulation and meshing of the geometrical entities used in **Vulcan**.

First, we will create a line. Next, we will save the project and it will be described in the **Vulcan** data base form. Starting from this line, we will create a square surface, which will be meshed to obtain a surface mesh. Finally, we will use this surface to create a cubic volume, from which a volume mesh can then be generated.

1. CREATION AND MESHING OF A LINE

We will begin the example creating a line by defining its origin and end points, points 1 and 2 in the following figure, whose coordinates are (0,0,0) and (10,0,0) respectively.
It is important to note that in creating and working with geometric entities, **Vulcan** follows the following hierarchical order: point, line, surface, and volume.



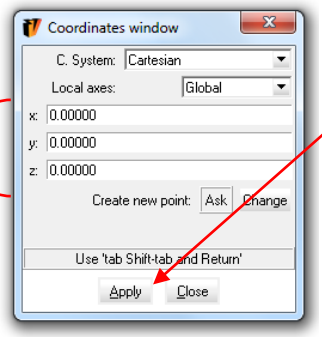
To begin working with the program, open **Vulcan**, and a new **Vulcan** project is created automatically.
From this new database, we will first generate points 1 and 2.

Next, we will create points 1 and 2. To do this, we will use an **Auxiliary Window** that will allow us to simply describe the points by entering coordinates.

Then, from the Top Menu, select **Geometry→Create→Point** and then select the sequence: **Utilities→Tools→Coordinates Window**

In the coordinate window opened previously, the following indicated steps should be used:

(1) Introduce the coordinates of point 1



(2) Create point 1 by clicking on the button **Apply** or by pressing **Enter** on the keyboard

And create point 2 in the same way, introducing its coordinates (10 0 0) in the **Coordinates Window**.

The last step in the creation of the points, as well as any other command, is to press **Escape**, either via the **Escape** button on the keyboard or by pressing the central mouse button. Select **Close** to close the **Coordinates Window** and go to **View→Zoom→Frame** in order to see the two points created.

Now, we will create the line that joins the two points. Choose from the **Top Menu**: **Geometry→Create→Straight line.** Option in the **Toolbar** shown below can also be used.

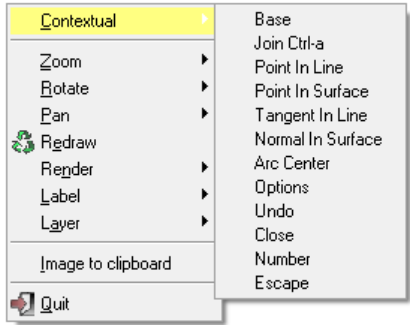
Next, the origin point of the line must be defined. In the **Mouse Menu**, opened by clicking the right mouse button, select **Contextual→Join C-a.**

NOTE: With option Join, a point already created can be selected on the screen. The command No Join is used to create a new point that has the coordinates of the point that is selected on the screen. We can see that the cursor changes form for the Join and No Join commands.

- Cursor during use of **Join** command
- Cursor during use of No **Join** command

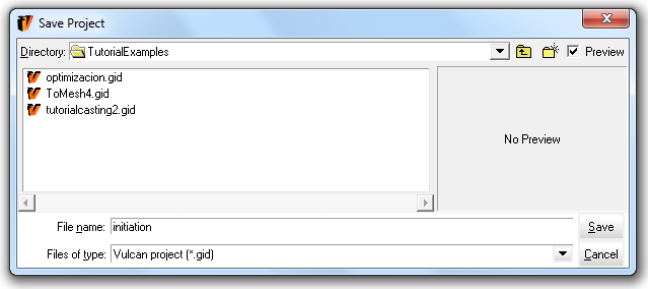
Now, choose on the screen the first point, and then the second, which define the line. Finally, press **Escape** to indicate that the creation of the line is completed.

NOTE: It is important to note that the Contextual submenu in the Mouse Menu will always offer the options of the command that is currently being used. In this case, the corresponding submenu for line creation has the following options:



Once the line has been generated, the project should be saved. To save the example select from the **Top Menu: Files→Save.**

The program automatically saves the file if it already has a name. If it is the first time the file has been saved, the user is asked to assign a name. For this, an **Auxiliary Window** will appear which permits the user to browse the computer disk drive and select the location in which to save the file. Once the desired directory has been selected, the name for the actual project can be entered in the space titled **File Name**.



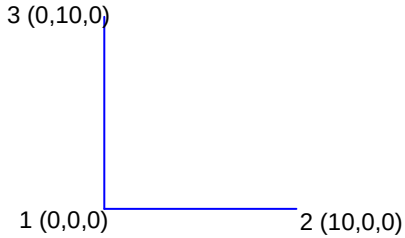
NOTE: Next, the manner in which **Vulcan** saves the information of a project will be explained. **Vulcan** creates a directory with a name chosen by the user, and whose file extension is **.gid**. **Vulcan** creates a set of files in this directory where all the information generated in the present example is saved. All the files have the same name of the directory to which they belong, but with different extensions. These files should have the name that **Vulcan** designates and should not be changed manually.

Each time the user selects option **save** the database will be rewritten with the new information or changes made to the project, always maintaining the same name.
To exit **Vulcan**, simply choose **Files→Quit.**

To access the example, **example.gid**, simply open **Vulcan** and select from the **Top Menu: Files→Open.** An **Auxiliary Window** will appear which allows the user to access and open the directory **iniciacion.gid**.

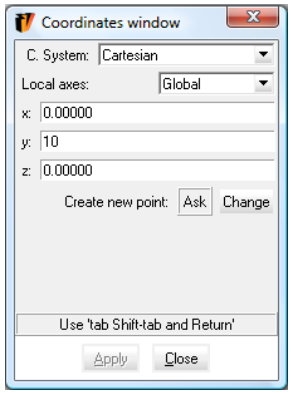
2. CREATION AND MESHING OF A SURFACE

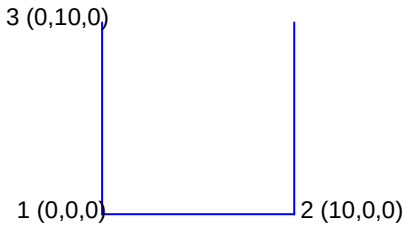
We will now continue with the creation and meshing of a surface.
First, we will create a second line between points 1 and 3.



We will now generate the second line. We will now use again the **Coordinates Window** to enter the points. (**Utilities→Tools→Coordinates Window**)

Select the line creation tool in the toolbar, select the point (0 0 0) with the option Join Ctrl-a and enter point (0,10,0) in the **Coordinates Window** and click **Apply**.



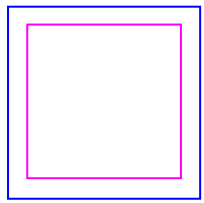


With this, a right angle of the square has been defined. If the user wants to view everything that has been created to this point, the image can be centered on the screen by choosing in the **Mouse Menu: Zoom→Frame**. This option is also available in the toolbar.

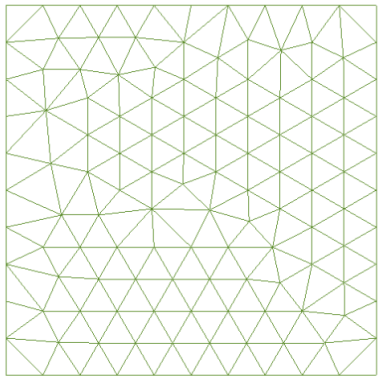
Finish the square by creating point (10,10,0) and the lines that join this point with points 2 and 3.

Now, we will create the surface that these four lines define. To do this, access the create surface command by choosing: **Geometry→Create→NURBS surface→By contour**. This option is also available in the toolbar:

Vulcan then asks the user to define the 4 lines that describe the contour of the surface. Select the lines using the cursor on the screen, either by choosing them one by one or selecting them all with a window. Next, press **Escape**.
As can be seen below, the new surface is created and appears as a smaller, magenta-colored square drawn inside the original four lines.



Once the surface has been created, the mesh can be created in the same way as was done for the line. From the **Top Menu** select: **Mesh→Generate mesh.**



An **Auxiliary Window** appears which asks for the maximum size of the element, in this example we define a size of 1.

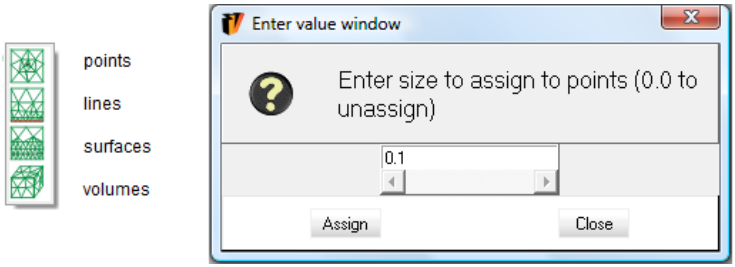
When the mesh it´s finished, if we want to see the mesh we have to select this option. This option allows to show or to hide the mesh.

We can see that the lines containing elements of two nodes have not been meshed. Rather the mesh generated over the surface consists of planes of three-nodded, triangular elements.

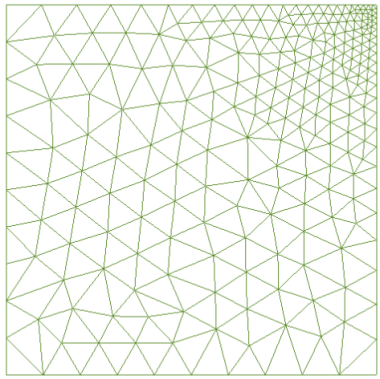
NOTE: Vulcan meshes by default the entity of highest order with which it is working.

Vulcan allows the user to concentrate elements in specified geometry zones. Next, a brief example will be presented in which the elements are concentrated in the top right corner of the square.

This operation is realized by assigning a smaller element size to the point in this zone than for the rest of the mesh. Select the following sequence: **Mesh→ Unstructured→Assign sizes on points.** The following dialog box appears, in which the user can define the size:



We enter the size, choose the right superior point, and press escape two times.
We must now regenerate the mesh (**Mesh→Generate Mesh**), canceling the mesh generated earlier, and we obtain the following:



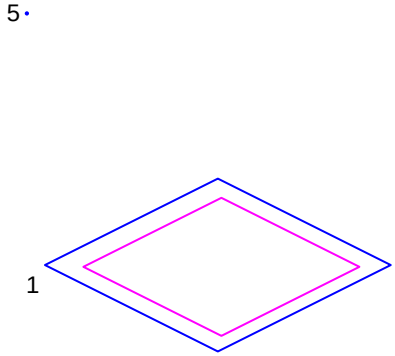
As can be seen in the figure above, the elements are concentrated around the chosen point. Various possibilities exist for controlling the evolution of the element size, which will be presented later in the manual.

3. CREATION AND MESHING OF A VOLUME

We will now present a study of entities of volume. To illustrate this, a cube and a volume mesh will be generated.
Without leaving the project, save the work done up to now by choosing **Files→Save,** and return to the geometry last created by choosing **Geometry→View geometry.**

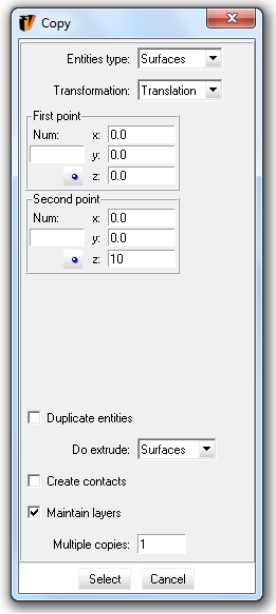
In order to create a volume from the existing geometry, firstly we must create a point that will define the height of the cube. This will be point 5 with coordinates (0,0,10), superimposed on point 1. (To view the new point, we must rotate the figure by selecting from the **Mouse Menu, Rotate→Trackball.** This option is also available in the toolbar:

Rotate the figure until the following position is achieved:



Next, we will create the upper face of the cube by copying from point 1 to point 5 the surface created previously. To do this, select the copy command, **Utilities→Copy.**

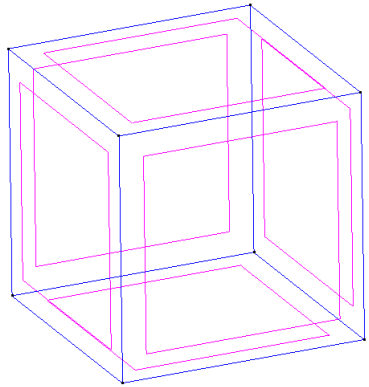
In the **Copy** window, we define the translation vector with the first and second points, in this case (0,0,0) and (0,0,10). Option **Do extrude surfaces** must be selected; this option allows us to create the lateral surfaces of the cube.



NOTE: If we look at the **Copy Window,** we can see an option called **Duplicate entities**. By activating this option, when the entities are copied (in this case from point 1 to point 5) **Vulcan** would create a new point (point 6) with the same coordinates as point 5.

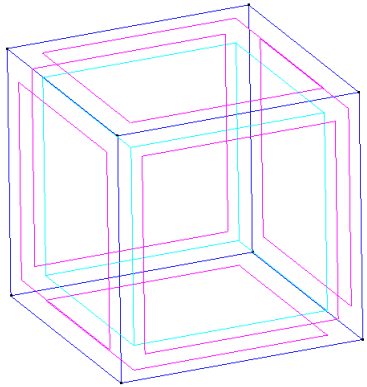
If the user does not choose option **Duplicate entities**, point 6 will be merged with point 5 when the entities are copied. By labeling the entities we could verify that only one point has been created.

Finishing the copy command for the surface, we obtain the following surfaces:

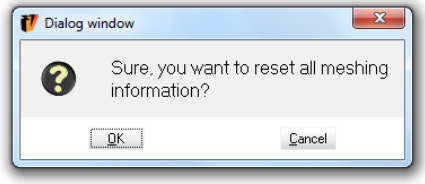


Now, we can generate the volume delimited by these surfaces. To create the volume, simply select the command **Geometry→Create→Volume→By contour**. This option is also available in the toolbar:

Select all the surfaces. **Vulcan** automatically generates the volume of the cube. The volume viewed on the screen is represented by a cube with an interior color of sky blue.



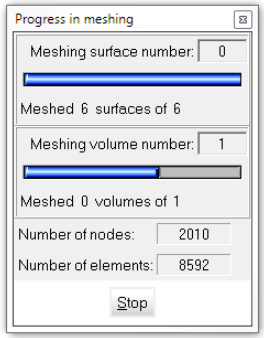
Before proceeding with the mesh generation of the volume, we should eliminate the information of the structured mesh created previously for the surface. Do this by selecting **Mesh→Reset mesh data,** and the following dialog box will appear on the screen:

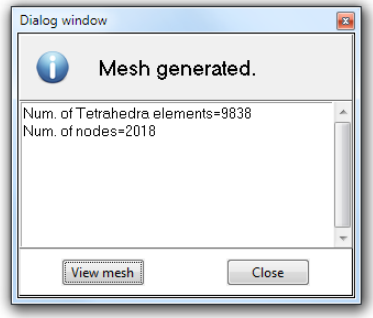


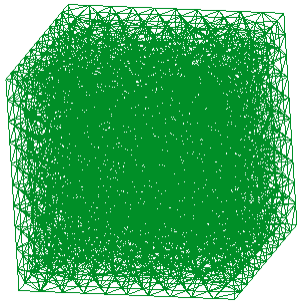
In which the user is asked to confirm the erasure of the mesh information.

NOTE: Another valid option would be to assign a size of 0 to all entities. This would eliminate all the previous size information as well as the information for the mesh, and the default options would become active.

Next, generate the mesh of the volume by choosing **Mesh→Generate mesh.** Another **Auxiliary Window** appears into which the size of the volumetric element must be entered. In this example, the value is 1.

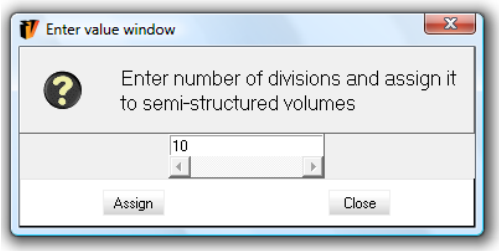


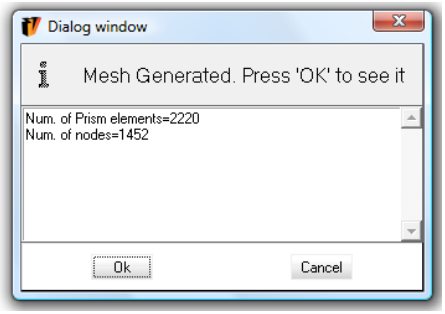


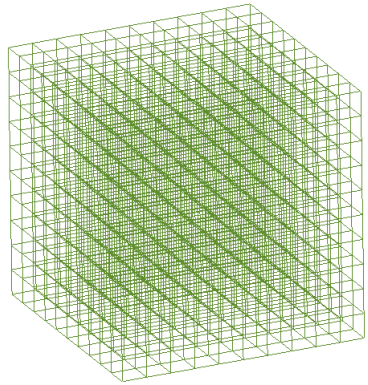


The mesh generated above is composed of tetrahedral elements of four nodes, but **Vulcan** also permits the use of hexahedral, eight-nodded structured elements.
We will generate a structured mesh of the volume of the cube. This is done by selecting in the right command bar: **Meshing→Structured→Volumes.** *Again, there are no structured meshes in casting problems. You can skip this step and continue with the next tutorial.*

Now select the volume to mesh and enter the number of partitions in its edges which will be created. Then, create again the mesh.







NOTE: Vulcan only allows the generation of structured meshes of 6-sided volumes.

With this example, the user has been introduced to the basic tools for the creation of geometric entities and mesh generation.

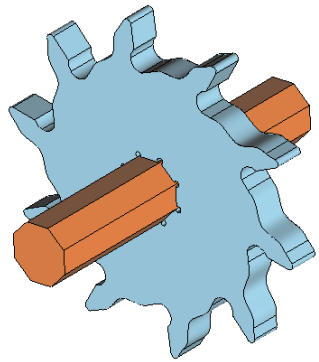
CASE STUDY 1

B. IMPLEMENTING A MECHANICAL PART

The objective of this case study is implementing a mechanical part in order to study it through meshing analysis. The development of the model consists of the following steps:

- Creating a profile of the part
- Generating a volume defined by the profile
- Generating the mesh for the part

At the end of this case study, the user should be able to handle the 2D tools available in VULCAN as well as the options for generating meshes and visualizing the prototype.



[This page is intentionally left blank]

1. WORKING BY LAYERS

1.1.Defining the layers

A geometric representation is composed of four types of entities, namely, points, lines, surfaces, and volumes.

A layer is a grouping of entities. Defining layers in computer-aided design permits us to work collectively with all the entities in one layer.

The creation of a profile of the mechanical part in our case study will be carrried out with the help of auxiliary lines. Two layers will be defined in order to prevent these lines from appearing in the final drawing. The lines that define the profile will be assigned to one of the layers, called the "profile" layer, while the auxiliary lines will be assigned to the other layer, called the "aux" layer. When the design of the part has been completed, the entities in the "aux" layer will be eliminated.

***NOTE**: You can find the finished model in the VULCAN CD-ROM.*

1.2.Creating two new layers

1. Open the layer management window. This is found in **Utilities→Layers**.
2. Create two new layers called "aux" and "profile." Select “layer0” and with the right button mouse select **Rename** and write the new name (aux). Then create a new layer called “profile” using the option **New layer** of the right button mouse and rename it as a profile.
3. Choose “aux” as the activated layer. To do this, doubleclick on "aux" to highlight it. From now on, all the entities created will belong to this layer.

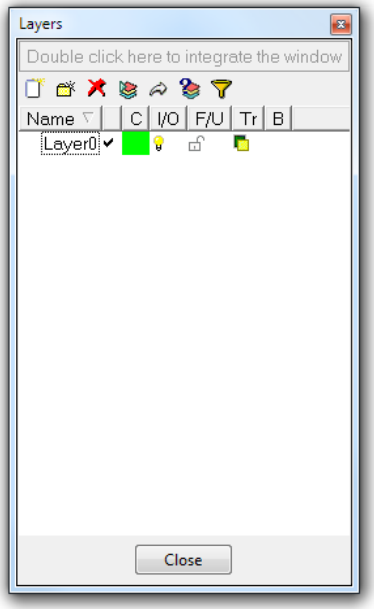


*Figure 1. The **Layers** window*

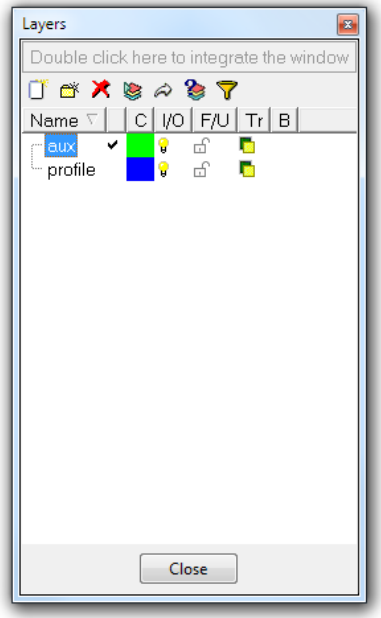


*Figure 2. The **Layers** window*

2. CREATING A PROFILE

In our case, the profile consists of various teeth. Begin by drawing one of these teeth, which will be copied later to obtain the entire profile.

2.1.Creating a size-55 auxiliary line

1. Choose the option **Line**, by going to **Geometry→Create→Straight Line** or by going to the VULCAN Toolbox[1].
2. Enter the coordinates of the beginning and end points of the auxiliary line[2]. For our example, the coordinates are (0, 0) and (55, 0), respectively. Besides creating a straight line, this operation implies creating the end points of the line.
3. Press **ESC**[3] to indicate that the process of creating the line is finished.
4. If the entire line does not appear on the screen, use the option **Zoom Frame,** which is located either in the VULCAN Toolbox or in **Zoom** on the mouse menu.

Figure 2. Creating a straight line

***NOTE**: The option **Undo**, located in **Utilities→Undo**, enables the user to undo the most recent operations. When this option is activated, a window appears in which to select all the operations to be undone.*

[1] The VULCAN Toolbox is a window containing the icons for the most frequently executed operations. For information on a particular tool, click on the corresponding icon with the right mouse button.

[2] The coordinates of a point may be entered on the command line with either a separation between them or a comma between them. If the Z coordinate 0 0is not entered, it is considered 0 by default. After entering the numbers, press **Return**. Another option for entering a point is using the **Coordinates Window**, found in **Utilities→Tools→Coordinates Window**.

[3] Pressing the **ESC** key is equivalent to pressing the center mouse button.

2.2. Dividing the auxiliary line near "point" (coordinates) (40, 0)

1. Choose **Geometry→Edit→Divide→Lines→Near Point.** This option will divide the line at the point ("element") on the line closest to the coordinates entered.
2. Enter the coordinates of the point that will divide the line. In this example, the coordinates are (40, 0). On dividing the line, a new point (entity) has been created.
3. Notice that the pointer has become a cross. Select the line that is to be divided by clicking on it.
4. Press **ESC** to indicate that the process of dividing the line is finished.

Figure 3. Division of the straight line near "point" (coordinates) (40, 0)

2.3. Creating a 3.8-radius circle around point (40, 0)

1. Choose the option **Geometry→Create→Object→Circle**.
2. The center of the circle (40, 0) is a point that already exists. To select it, go to **Contextual→Join C-a** on the mouse menu (right button). The pointer will become a cross, which means that you may click on the point.
3. Enter any point that, together with the center of the circle, defines a normal to the XY plane, i.e., (0, 0, 40).
4. Enter the radius of the circle. The radius is 3.8[4]. Two circumferences are created; the inner circumference represents the surface of the circle.
5. Press **ESC** to indicate that the process of creating the circle is finished.

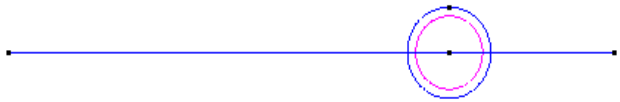


Figure 4. Creating a circle around a point (40, 0)

[4] In VULCAN the decimals are entered with a point, not a comma.

2.4.Rotating the circle -3 degrees around a point

1. Use the **Move** window, which is located in **Utilities→Move**.
2. Within the **Move** menu and from among the **Transformation** possibilities, select **Rotation**. The type of entity to receive the rotation is a surface. Therefore, from the menu **Entities Type**, choose **Surfaces**.
3. Enter -3 in the **Angle** box and click a checkmark into the box preceding **Two dimensions.** (Provided we define positive rotation in the mathematical sense, which is counterclockwise, -3 degrees will mean a clockwise rotation of 3 degrees.)
4. Enter the point (0, 0, 0) under **First Point**. This is the point that defines the center of rotation.
5. Click **Select** to select the surface that is to rotate, which in this case is that of the circle.
6. Press **ESC** (or **Finish** in the **Move window**) to indicate that the selection of surfaces to rotate has been made, thus executing the rotation.

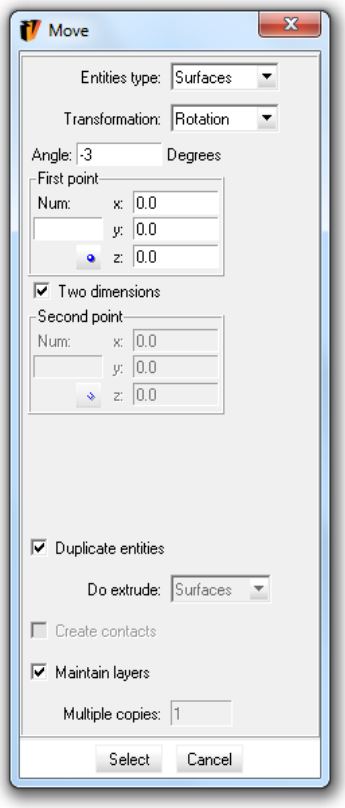


*Figure 5. The **Move** window*

2.5. Rotating the circle 36 degrees around a point and copying it.

1. Use the **Copy** window, located in **Utilities→Copy**.
2. Repeat the rotation process explained in section 2.4, but this time with an angle of 36 degrees. (See Figure 6.)

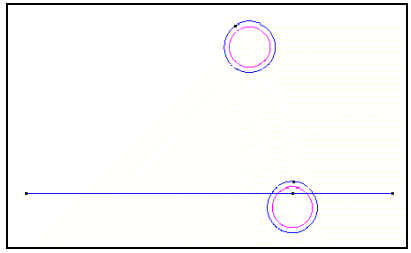


Figure 6. Result of the rotations

***NOTE**: The **Move** and **Copy** windows differ only in that **Copy** creates new entities but **Move** only displaces entities already selected.*

2.6. Rotating and copying the auxiliary lines

1. Use the **Copy** window, located in **Utilities→Copy.** (See Figure 9.)
2. Repeat the rotating and copying process from section 2.5 for the two auxiliary lines. Select the option **Lines** from the **Entities type** menu and enter an angle of 36 degrees.
3. Select the lines to copy and rotate. Do this by clicking **Select** in the **Copy** window.
4. Press **ESC** to indicate that the process of selecting is finished, thus executing the task. (See Figure 7.)

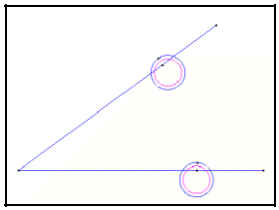


Figure 7. Result of copying and rotating the line.

5. Rotate the line segment that goes from the origin to point (40, 0) an angle of 33 degrees and copy it. (See Figure 8.)

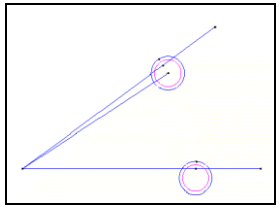


Figure 8. Result of the rotations and copies

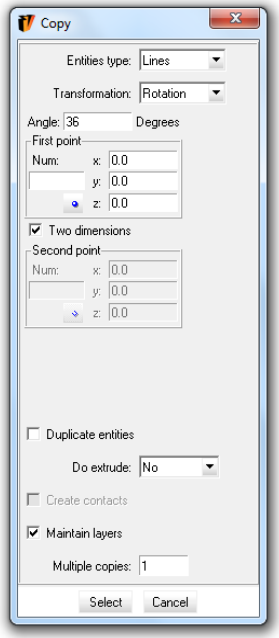


Figure 9. The **Copy** window

***NOTE**: In the **Copy** and **Move** windows, the option **Pick** may be used to select existing points with the mouse.*

2.7. Intersecting lines

1. Choose the option **Geometry→ Edit→Intersection→Lines.**

2. Select the upper circle resulting from the 36-degree rotation executed in section 2.5.

3. Select the line resulting from the 33-degree rotation executed in section 2.6. (See Figure 10). The intersection has created a point (Figure 11)

4. Press **ESC** to conclude the intersection of lines.

5. Create a line between point (55, 0) and the point generated by the intersection. To select the points, use the option **Join C-a** in the tool **Line**.

6. Choose the option **Geometry→ Edit→Intersection→Lines** to make another intersection between the lower circle and the line segment between point (40, 0) and point (55, 0). (See Figure 12.)

7. Again, go to the option **Geometry → Edit→Intersection→Lines** to make an intersection between the upper circle and the farthest segment of the line that was rotated 36 degrees. (See Figure 12.)

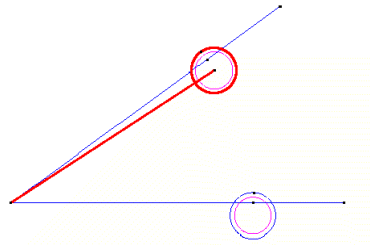


Figure 10. The two lines selected

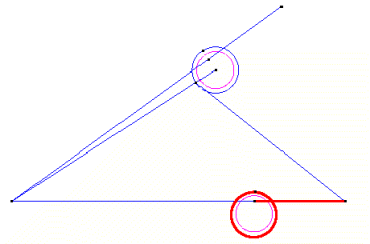


Figure 11. Intersecting lines

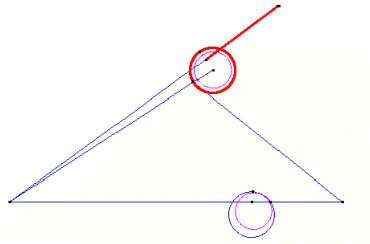


Figure 12. Intersecting lines

NOTE: The order of selection in the intersecting process is important since the second line selected can not have higher entities.

2.8. Creating and arc tangential to two lines

1. Choose **Geometry→Create→Arc→By3points** (or go to the VULCAN Toolbox).
2. Open the mouse menu and go to **Contextual**, click **By Tangents.** Enter a radius of 1.35 on the command line (see footnote 2 on page 4).

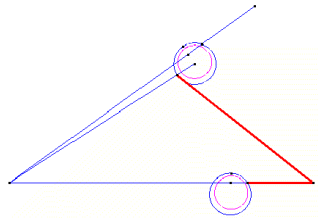


Figure 13. The line segments to be selected

3. And select the two line segments shown in Figure 13. Then press **ESC** to indicate that the process of creating the arcs is finished.

2.9. Translating the definitive lines to the "profile" layer

1. If the "profile" layer is not already selected, doubleclick on it to select it.
2. Select the "profile" layer in the **Layers** window. The auxiliary lines will have been eliminated and the "profile" layer will contain only the definitive lines.
3. Click the right button mouse over the layer profile and select **Send To,** choose **Lines** in order to select the lines to be translated. Select only the lines that form the profile (Figure 14). To conclude the selection process, press the **ESC** key or click **Finish** in the **Layers** window.

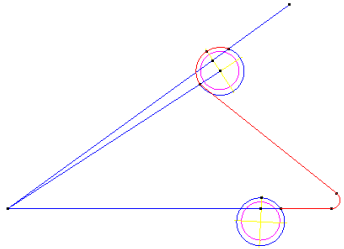


Figure 14. Lines to be selected

2.10. Deleting the "aux" layer

1. Click **Off** the profile layer.
2. Choose **Geometry→Delete→All Types** (or use the VULCAN Toolbox).
3. Select all the lines that appear on the screen. (The click-and-drag technique may be used to make the selection.)
4. Press **ESC** to conclude the selection of elements to delete.
5. Select the "aux" layer in the **Layers** window and click **Delete**.

NOTE: *When a layer is clicked **Off**, VULCAN gives note of this. From that moment on, whatever is drawn does not appear on the screen since it goes on the hidden layer.*

NOTE: *To cancel the deletion of elements after they have been selected, open the mouse menu, go to **Contextual** and choose **Clear Selection**.*

NOTE: *Elements forming part of higher level entities may not be deleted. For example, a point that defines a line may not be deleted.*

NOTE: *A layer containing information may not be deleted. First the contents must be deleted.*

2.11.Rotating and obtaining the final profile

1. Make sure that the activated layer is the "profile" layer. (By dobleclick).
2. In the **Copy** window, select the line rotation (**Rotation**, **Lines**).
3. Enter an angle of 36 degrees. Make sure that the center is point (0, 0, 0) and that you are working in two dimensions.
4. In the option **Multiple Copies** enter 9. This way, 9 copies will be made, thus obtaining the 10 teeth that form the profile of the model (9 copies and the original).
5. Highlight **Select** and select the profile. To conclude the operation, press the **ESC** key or click **Finish** in the **Copy** window. The result is shown in Figure 15.

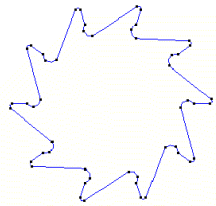


Figure 15. The part resulting from this process

2.12. Creating a surface

1. Create a NURBS surface. To do this, select the option **Geometry→ Create→ NURBS Surface→By Contour.** This option can also be found in the VULCAN Toolbox.
2. Select the lines that define the profiel of the part and press **ESC** to create the surface.
3. Press **ESC** again to exit the function. The result is shown in Figure 16.

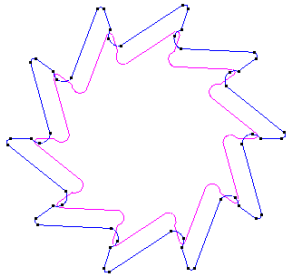


Figure 16. Creating a surface starting from the contour

***NOTE**: To create a surface there must be a set of lines that define a closed contour.*

3. CREATING A HOLE IN THE PART

In the previous sections we drew the profile of the part and created the surface. In this section we will make a hole, an octagon with a radius of 10 units, in the surface of the part. First we will draw the octagon.

3.1. Creating two sides of the octagon

1. Create a point (10, 0). Choose **Geometry→Create→Point** and enter the coordinates in the command bar. Press **ESC** to conclude the insertion of the point.

2. In the **Copy** window, select **Points** and **Rotation**. Enter an angle of 45 degrees and select the **two dimensions** option. In the option **Multiple Copies** enter 2.

3. Select the tool **Line**. Select three consecutive points to create two sides of the octagon with **Join C-a** located in **Contextual** on the mouse menu. Press **ESC** to close the tool **Line**. See Figure 17.

Figure 17. Creating the first quadrant

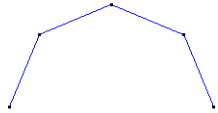


Figure 18. Symmetry relative to the vertical axis

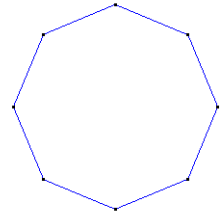


Figure 19. Symmetry relative to the horizontal axis

3.2. Creating the rest of the octagon by mirror effect

1. In the **Copy** window, choose **Lines** and **Mirror**. Be sure the option **Two Dimensions** has been selected.

2. Enter two points that define a vertical axis of symmetry, for example (0, 0, 0) and (0, 10, 0).

3. Choose **Select** and select the two sides of the octagon that have been drawn. Press **ESC** to conclude the selection. (Figure 18)

4. Repeat the process entering two points that define a horizontal axis of symmetry, for example (0, 0, 0) and (10, 0, 0). This time select the four sides of the octagon (Figure 19)

3.3. Creating a hole in the surface of the mechanical part

1. Choose the option **Geometry→Edit→Hole NURBS Surface**.
2. Select the surface in which to make the hole. (Figure 20)
3. Select the lines that define the hole (Figure 21) and press **ESC**.

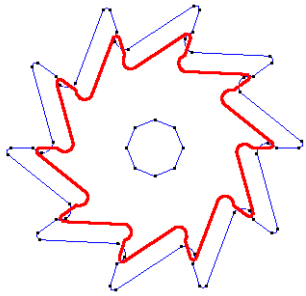


Figure 20. The selected surface in which to create the hole

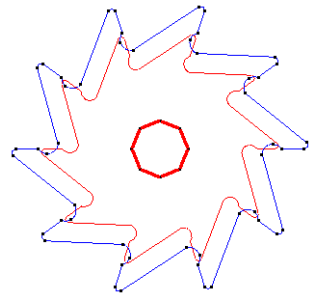


Figure 21. The selected lines that define the hole

4. Again, press **ESC** to exit this function.

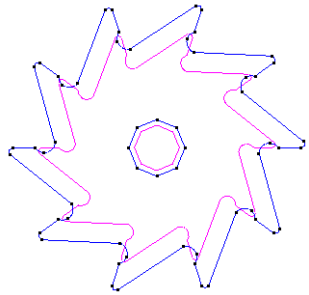


Figure 22. The model part with the hole in it

4. CREATING VOLUMES FROM SURFACES

The mechanical part to be constructed is composed of two volumes: the volume of the wheel (defined by the profile), and the volume of the axle, which is a prism with an octagonal base that fits into the hole in the wheel. Creating this prism will be the first step of this stage. It willl be created in a new layer that we will name "prism."

4.1. Creating the "prism" layer and translating the octagon to this layer

1. In the **Layers** window, create a new layer and rename as a “prism”.
2. Select the "prism" layer and doubleclick to configure it as the activated layer.
3. With the right button mouse over the layer prism choose **send to/ Lines**. Select the lines that define the octagon. Press **ESC** to conclude the selection.

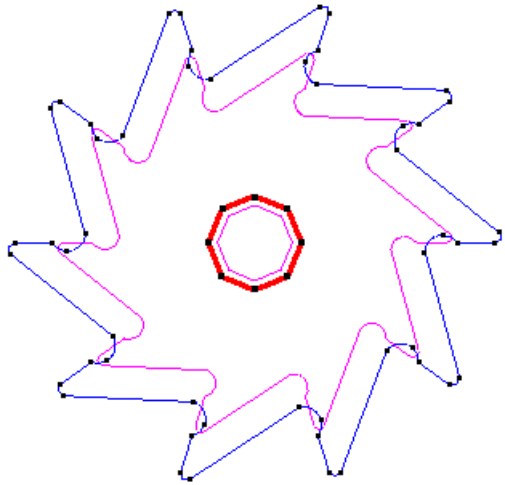


Figure 23. The lines that form the octagon

4. Select the "profile" layer and click **Off** to deactivate it → . And doubleclick again over the layer “prism”.

4.2. Creating the volume of the prism

1. First copy the octagon at a distance of -50 units relative to the surface of the wheel, which is where the base of the prism will be located. In the **Copy** window, choose **Translation** and **Lines**. Since we want to translate 50 units, enter two points that define the vector of this translation, for example (0, 0, 0) and (0, 0, 50).
2. Choose **Select** and select the lines of the octagon. Press **ESC** to conclude the selection.

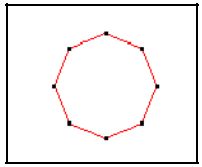


Figure 24. Selection of the lines that form the octagon

3. Since the Z axis is parallel to the user's line of vision, the perspective must be changed to visualize the result. To do this, use the tool **Rotate Trackball**, which is located in the VULCAN Toolbox or on the mouse menu.

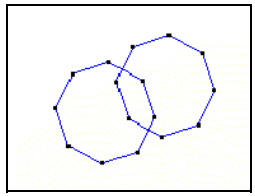


Figure 25. Copying the octagon and changing the perspective

4. Choose **Geometry→Create→Nurbs surface→By contour**. Select the lines that form the displaced octagon and press **ESC** to conclude the selection. Again, press **ESC** to exit the function of creating the surfaces.

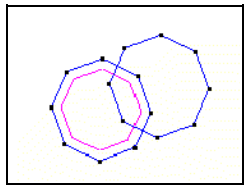


Figure 26. The surface created on the translated octagon

5. In the **Copy** window, choose **Translation** and **Surfaces**. Make a translation of 110 units. Enter two points that define a vector for this translation, for example (0, 0, 0) y (0, 0, -110).
6. To create the volume defined by the translation, select **Do Extrude Surfaces** in the **Copy** window.
7. Click **Select** and select the surface of the octagon. Press **ESC**. The result is shown in Figure 27.

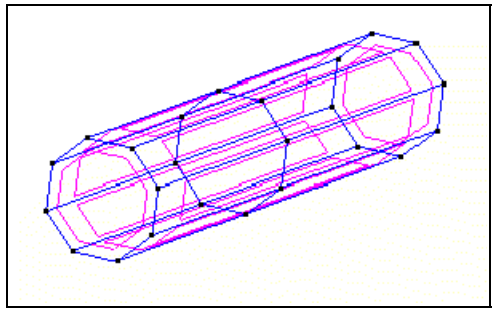


Figure 27. The result of the extrusion

8. Choose **Geometry→Create→Volume→By contour**. Select all the surfaces that form the prism and press **ESC**.
9. Again, press **ESC** to exit the function of creating volumes

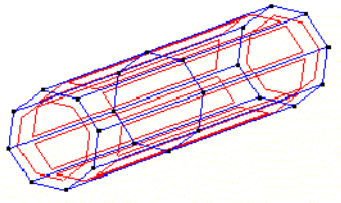


Figure 28. Selection of the surfaces of the prism

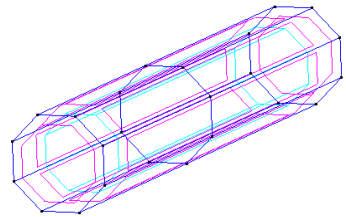


Figure 29. Creation of the volume of the prism

10. Choose the option **Render→Flat** from the mouse menu to visualize a more realistic version of the model. Then return to the normal visualization using **Render→ Normal**.

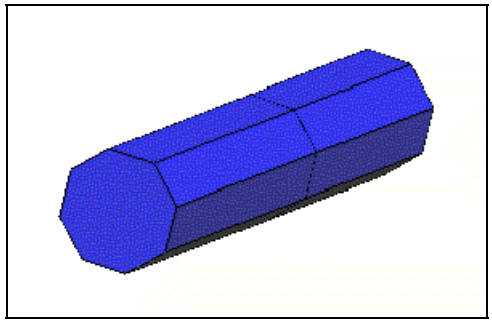


*Figure 30. Visualization of the prism with the option **Render→Flat**.*

***NOTE**: In the option **Color** in the **Layers** window, the user may define the color of the selected layer. The color of the layer is that used in the rendering.*

4.3. Creating the volume of the wheel

1. Deactivate the "prism" layer. → Visualize the "profile" layer and activate it (doble click on it). The volume of the wheel will be created in this layer.
2. In the **Copy** window, choose **Translation** and **Surfaces**. A translation of 10 units will be made. To do this, enter two points that define a vector for this translation, for example (0, 0, 0) and (0, 0, -10).
3. Choose the option **Do Extrude Volume** from the **Copy** window. The volume that is defined by the translation will be created.
4. Click **Select** and select the surface of the wheel. Press **ESC**.
5. Select the two layers and click them **On** so that they are visible.
6. Choose **Render→Flat** from the mouse menu to visualize a more realistic version of the model (Figure 31).

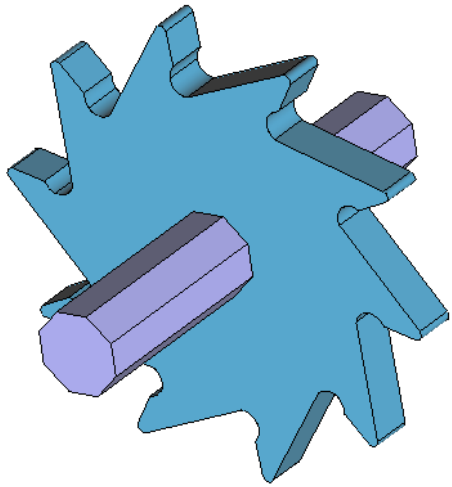


Figure 31. Image of the wheel

5. GENERATING THE MESH

Now that the part has been drawn and the volumes created, the mesh may be generated. First we will generate a simple mesh by default.

Depending on the form of the entity to mesh, VULCAN makes an automatic correction of the element size. This correction option, which by default is activated, maybe modified from the **Preferences** window, in **Meshing**, under the name of **Automatic correct sizes**. Automatic correction is sometimes not sufficient. In such cases, the user must indicate where a more precise mesh is needed. Thus, in this example, we will increase the concentration of elements along the profile of the wheel by following two methods: 1) assigning element sizes around points and 2) assigning element sizes around lines.

5.1. Generating the mesh by default

1. Choose **Mesh→Generate Mesh**.
2. A window comes up in which to enter the maximum element size of the mesh to be generated (Figure 32). Leave the default value given by VULCAN unaltered and click **OK**.

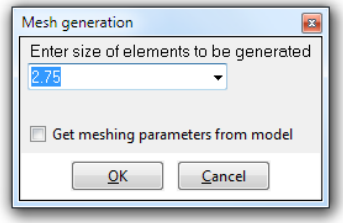


Figure 32. The window in which to enter the maximum element size

3. A window appears that shows the meshing progression. Once the process is finished, another window comes up with information about the mesh that has been generated (Figure 33). Click **View mesh** to visualize the resulting mesh (Figure 34).

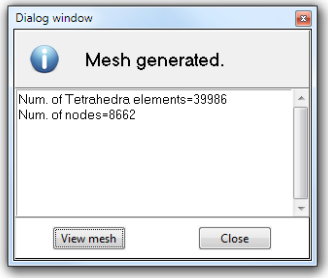


Figure 33. The window with information about the mesh generated

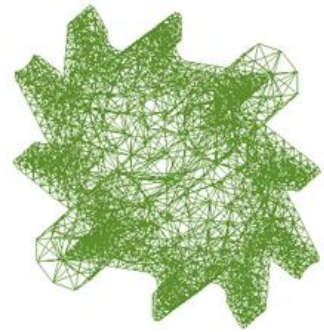


Figure 34. The mesh generated by default

4. Use the option **Mesh→View mesh boundary** to see only the contour of the volumes meshed without their interior (Figure 35). This mode of visualization may be combined with the various rendering methods. Use this button to toogle between the geometry and the mesh.

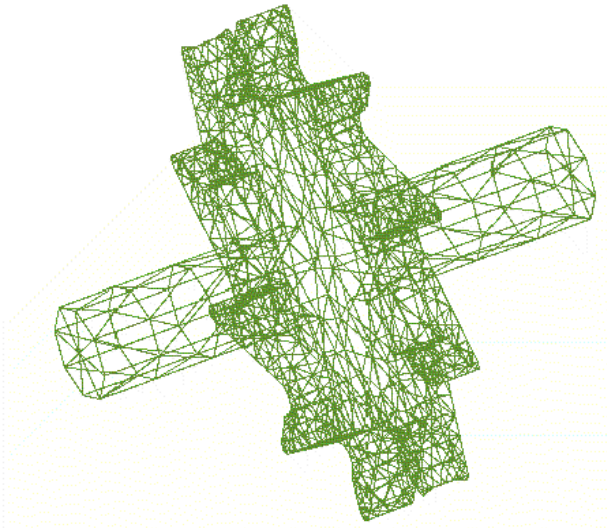


*Figure 35. Mesh visualized with the **Mesh→View mesh boundary** option*

5. Visualize the mesh generated with the various rendering option in the **Render** menu, located on the mouse menu.

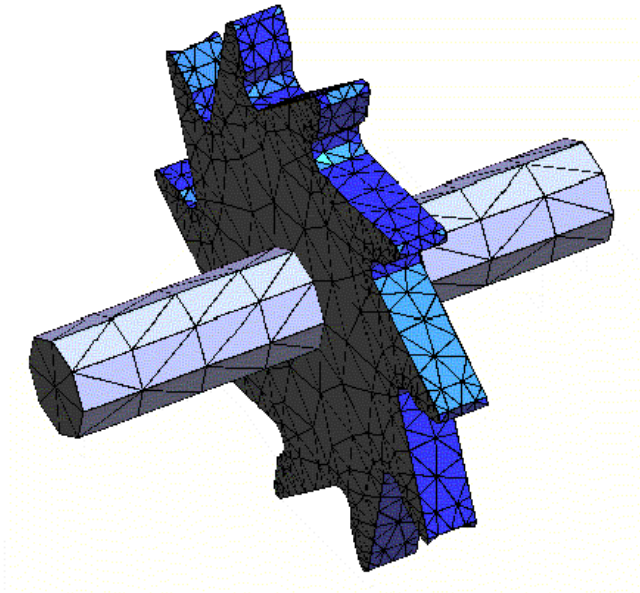


*Figure 36. Mesh visualized with **Render→Flat** combined with **Render→Normal**.*

5.2. Generating the mesh with assignment of size around points

1. Enter **rotate angle -90 90**[5] on the command line. This way we will have a side view.

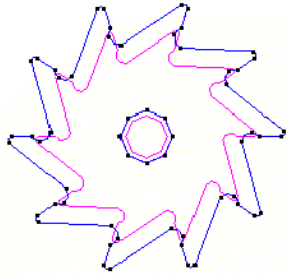


Figure 37. Side view of the part.

[5] Another option equivalent to **rotate angle -90 90 is Rotate→Plane XY**, located on the mouse menu.

2. Choose **Mesh→Unstructured→Assign sizes on points**. A window comes up in which to enter the element size around the point to be selected. Enter 0.7.
3. Select only the points on the wheel profile. One way of doing this is to select the entire part and then cancel the selection of the points that form the prism hole. Press **ESC** two times to conclude the selection process.

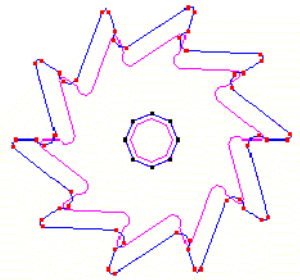


Figure 38. The selected points of the wheel profile

4. Choose **Mesh→Generate mesh**.

5. A window comes up asking if the previous mesh should be eliminated (Figure 39). Click **Yes**. Another window appears in which to enter the maximum element size. Leave the default value unaltered.

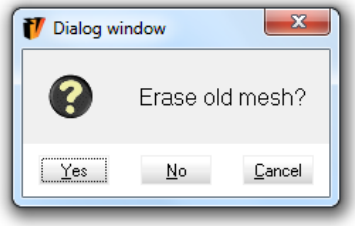


Figure 39

6. A third window shows the meshing process. Once it has finished, click **View mesh** to visualize the resulting mesh.

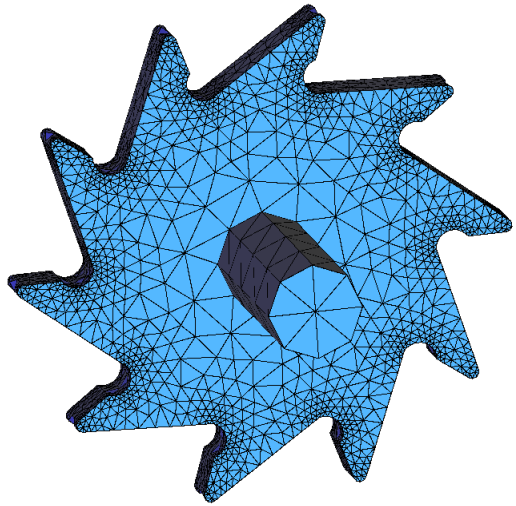


Figure 40. Mesh with assignment of sizes around the points on the wheel profile

7. A greater concentration of elements has been achieved around the points selected.
8. Choose **Geometry→View Geometry** to return to the normal visualization.

5.3. Generating the mesh with assignment of size around lines

1. Open the **Preferences** window, which is found in **Utilities**. Bring forward the **Meshing** card. In this window there is an option called **Unstructured Size Transitions**, which defines the size gradient of the elements. A high gradient number enables the user to concentrate more elements on the wheel profile. To do this, select a gradient size of 0.8. Click **Accept** and **close**.
2. Choose **Mesh→Reset mesh data** to delete the previously assigned sizes from section 5.2.
3. Choose **Mesh→Unstructured→Assign sizes on lines**. A window comes up in which to enter the element size around the lines to be selected. Enter size 0.7. Select only the lines of the wheel profile using the same process as in section 5.2.

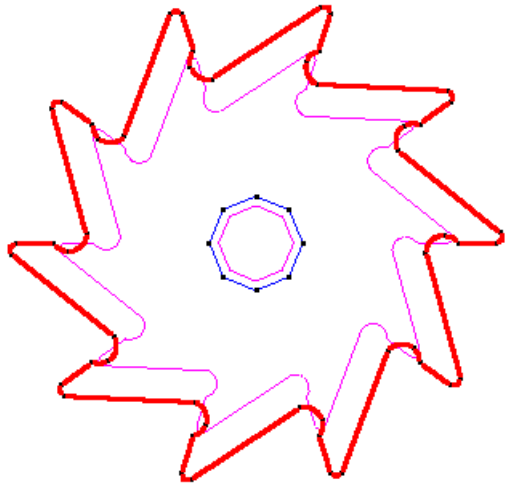


Figure 41. Selected lines of the wheel profile

4. Choose **Mesh→Generate mesh**. A window appears asking if the previous mesh should be eliminated. Click **Yes**.
5. Another window comes up in which to enter the maximum element size. Leave the default value unaltered.
6. A greater concentration of elements has been achieved around the lines selected. In contrast to the case in section 5.2., this mesh is more accurate since lines define the profile much better than points do (Figure 42).

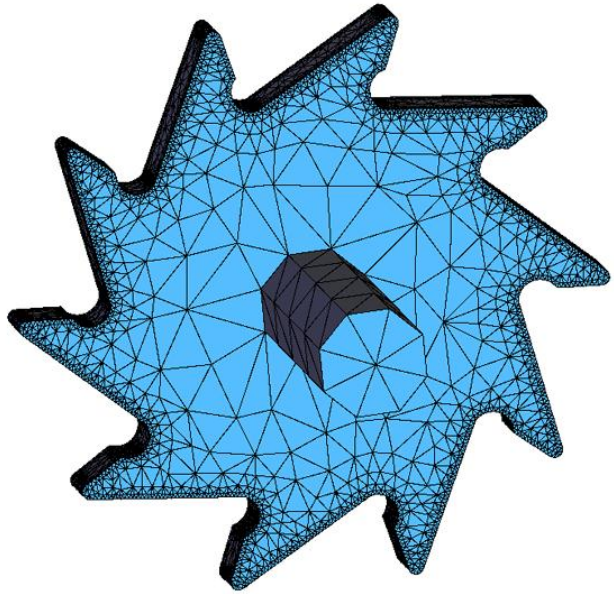


Figure 42. Mesh with assignment of sizes around lines

6. OPTIMIZING THE DESIGN OF THE PART

The part we have designed can be optimized, thus achieving a more efficient product. Given that the part will rotate clockwise, reshaping the upper part of the teeth could reduce the weight of the part as well as increase its resistance. We could also modify the profile of the hole in order to increase resistance in zones under axe pressure.

To carry out these optimizations, we will use new tools such as NURBS lines. The final steps in this process will be generating a mesh and visualizing the changes made relative to the previous design.

This example begins with a file named "optimizacion.gid". This file can be downloaded from the VULCAN web page http://www.quantech.es/support-vulcan.aspx

6.1. Modifying the profile

1. Choose the option **Open** from the **Files** menu and open the file "optimizacion.gid".
2. The file contents appears on the screen. In order to work more comfortably, select **Zoom In,** thus magnifying the image. This option is located both in the VULCAN Toolbox or in the mouse menu under **Zoom**.

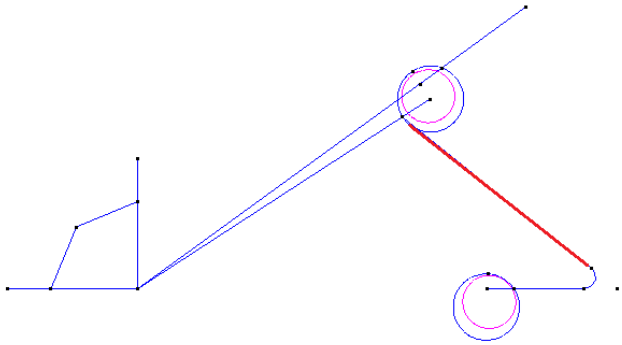


Figure 43. Contents of the file "optimizacion.gid".

3. Make sure that the "aux" layer is activated (doubleckick).
4. Choose **Geometry→Edit→Divide→Lines→Num Divisions.** This option divides a line into a specified number of segments
5. A window comes up in which to enter the number of partitions (8).
6. Select the line segment from the upper part of a tooth (Figure 43).
7. Using the option **Geometry→Create→Point,** create a point at the coordinates (40, 8.5).
8. Choose **Geometry→Create→NURBS line** to create a NURBS curve. The NURBS line to be created will pass through the two first points which have been created on dividing the line at point (40 8.5) and by the two last points of the divided line.

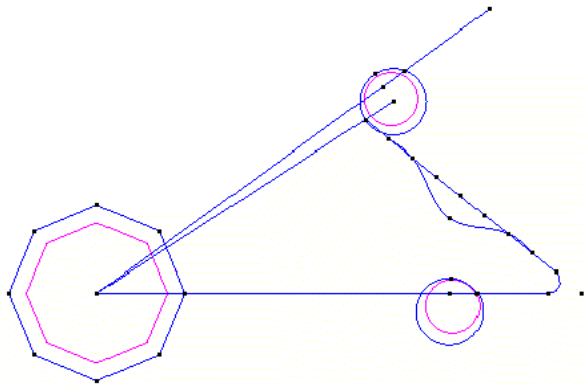


Figure 44. Optimizing the design

9. Select the first point through which the curve will pass. To do this use **Join C-a**, located in **Contextual** on the mouse menu.
10. One at a time, select the rest of the points except the last one. Use **Join C-a** each time in order to ensure that the line passes through the point.
11. Before selecting the last point, choose **Last Point** in **Contextual** on the mouse menu. Then finish the NURBS line. The result is shown in Figure 44.
12. Translate the new profile to the "profile" layer and eliminate the auxiliary lines and his points. (see the figure 45)
13. Repeat the process explained in section 2.11 to create the wheel surface. And, using **Geometry→ Create→NURBS Surface→By contour**, select it to create a NURBS surface

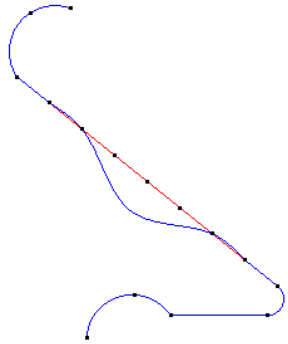


Figure 455. Optimizing the design

6.2. Modifying the profile of the hole

1. From now on we will work with the layer **aux.** Select like a layer to use (double click).
2. Choose **Geometry→Create→Object→Circle**. Enter (-10.5, 0) as the center point. Enter a normal to the **XY** plane (Positive Z) and a radius of 1.
3. With the tool **Geometry→Delete→Surface** in the Toolbox, delete the surface of the circle so that only the line is left. This way the option **Geometry→Edit→ Intersection→Lines** may be used to intersect the circle (circumference). Select only the circle and the two straight lines that intersect it.
4. Choose **Copy** from the **Utilities** menu and make two copies, rotating the circle -45 degrees.
5. Using the intersection options, delete the auxiliary lines leaving only the valid lines, thus obtaining a quarter of the profile of the hole. The result is illustrated in Figure 46.
6. In the **Copy** window, choose **Mirror** and **Lines**. Make sure that the option **Two Dimensions** is highlighted. Complete the mirror process obtaining a new hole (this process was explained in **section 3.2.** Delete the auxiliary lines.
7. Send to this lines to the layer "**profile**" and delete the layer "**aux**"Create the hole in the surface of the wheel using **Geometry→Edit→Hole NURBS Surface**.

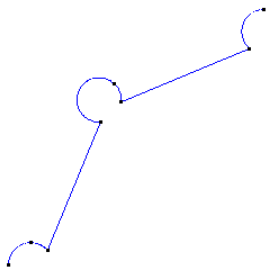


Figure 466. A quarter of the new hole profile

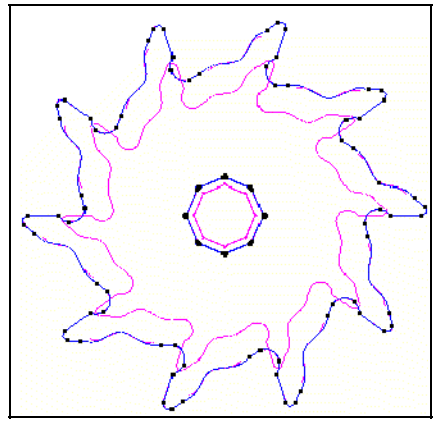


Figure 477. The surface of the new optimized design

6.3. Creating the volume of the new design

Repeat the same process as in section 4.3:

1. In the **Copy** window, choose **Translation** and **Surfaces**. Enter two points that define a translation of 10 units, for example (0, 0, 10) and (0, 0, 0).
2. Choose **Do Extrude Surfaces** in the **Copy** window.
3. Click **Select** and select the surface of the wheel. Press **ESC**.
4. Choose **Geometry→Create→Volume→By contour** and select all the surfaces of the part.

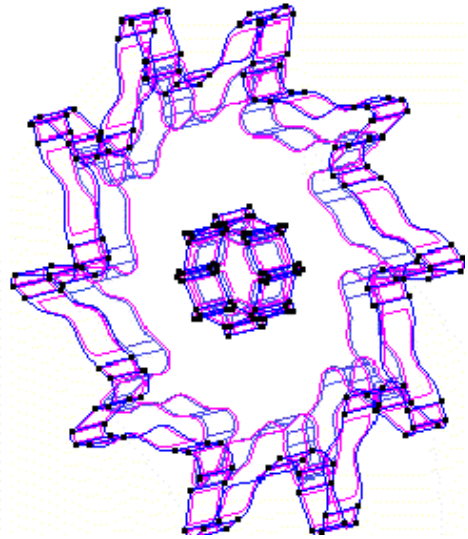


Figure 488. The volume of the optimized design

GENERATING THE MESH FOR THE NEW DESIGN

Generating the mesh for the optimized design is more complex. In this geometry it is especially important to obtain a precise mesh on the surfaces around the hole and on the surfaces of the teeth.

Initially, we will generate a simple mesh by default. Then we will generate a mesh using Chordal Error[6] to obtain a more accurate mesh.

7.1. Generating a mesh for the new design by default

1. Choose the option **Mesh→Generate Mesh**.
2. A window comes up in which to enter the maximum element size for the mesh to be generated. Leave the default value provided by VULCAN unaltered and click **OK**.
3. Once the mesh generating process is finished, select the icon to visualize the result (Figure 49).

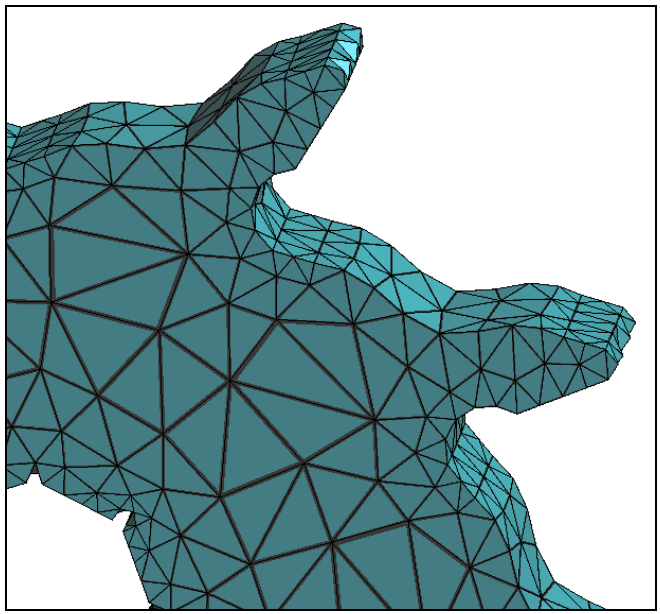


Figure 499. A detail of the mesh generated by default

[6] The Chordal Error is the distance between the element generated by the meshing process and the real profile.

7.2. Generating a mesh using "Chordal Error"

1. Choose **Mesh→Unstructured→Sizes by Chordal error**.
2. Now the maximum element size must be entered. Enter 15 on that command line.
3. The next step is entering the **chordal error**. Enter 0.05.
4. Choose **Mesh→Generate Mesh**
5. A greatly improved approximation has been achieved in zones containing curves and, more specifically, along the wheel profile and the profile of the hole. (See Figure 50.)

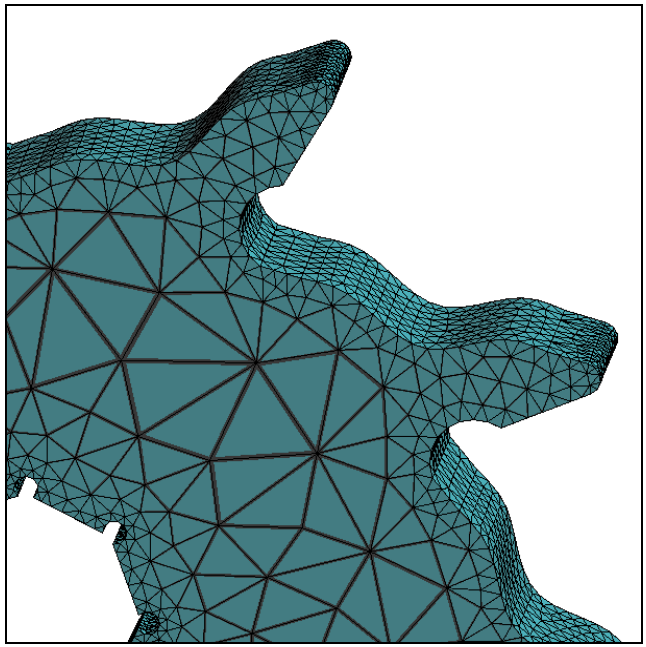


Figure 50. A detail of the mesh generated using Chordal Error

CASE STUDY 2

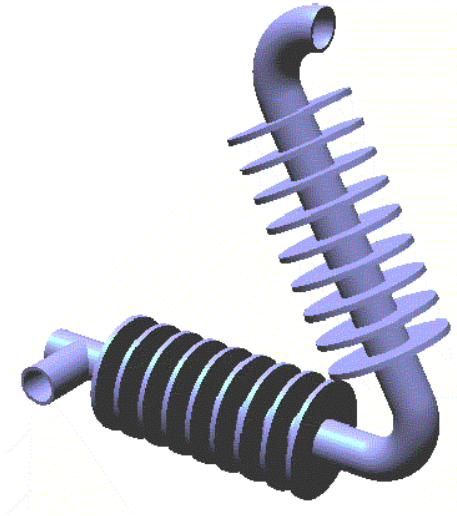
C. IMPLEMENTING A COOLING PIPE

This case study shows the modeling of a more complex piece and concludes with a detailed explanation of the corresponding meshing process. The piece is a cooling pipe composed of two sections forming a 60-degree angle.

The modeling process consists of four steps:

- Modeling the main pipes
- Modeling the elbow between the two main pipes, using a different file
- Importing the elbow to the main file
- Generating the mesh for the resulting piece

At the end of this case study, the user should be able to use the CAD tools available in VULCAN as well as the options for generating meshes and visualizing the result.



[This page is intentionally left blank]

1. WORKING BY LAYERS

Various auxiliary lines will be needed in order to draw the part. Since these auxiliary lines must not appear in the final drawing, they will be in a layer apart from the one used for the finished model.

1.1. Creating two new layers

1. Open the layer management window, which is found in the **Utilities→Layers** menu.

2. Create two new layers called "**aux**" and "**ok**". Enter the name for each layer in the Layers window (Figure 1).

3. Choose "aux" as the activated layer. (DoubleClick)

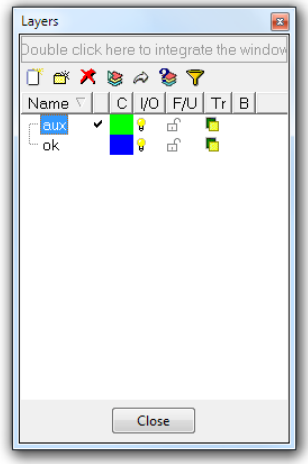


*Figure 1. The **Layers** window*

2. CREATING THE AUXILIARY LINES

The auxiliary lines used in this project are those that make it possible to determine the center of rotation and the tangential center, which will be used later to create the model.

2.1. Creating the axes

STEP 1

1. Choose the option **Line**, which is located in **Geometry→ Create→Line**[1].
2. Enter the coordinate (0, 0) on the command line.
3. Enter the coordinate (200, 0) on the command line.
4. Press **ESC**[2] to indicate that the process of creating the line is finished.
5. Again, choose **Line**. Draw a line between points (0, 25) and (200, 25). The result is shown in Figure 2.

Figure 2

6. Go to the **Copy** window (Figure 4), which is found in **Utilities→Copy**.
7. Choose **Rotation** from the **Transformation** menu and **Lines** from the **Entities Type** menu.
8. Enter an angle of -60 degrees and click on **Two dimensions**.

[1] This option is also found in the VULCAN Toolbox.
[2] Pressing the **ESC** key is equivalent to pressing the center mouse button.

9. Enter point (200, 0, 0) in **First Point**. This is the point that defines the center of rotation.

10. Click **Select** to select the first line to be drawn.

11. After making the selection, press **ESC** (or **Finish** in the **Move** window) to indicate that the selection of lines to rotate is finished. The result is shown in Figure 3.

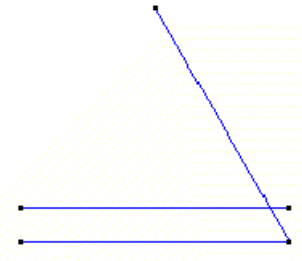


Figure 3. Creating the axes

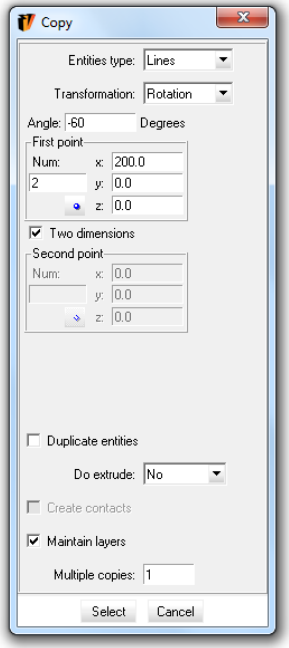


*Figure 4. The **Copy** window*

2.2. Creating the tangential center

STEP 2

1. Choose the option **Line**, located in **Geometry→ Create→Line.** On the mouse menu, choose **Contextual** and use **Join C-a** (or tap Ctrl + A) to select points (0, 0) and (0, 25). Press **ESC.**
2. In the **Copy** window, choose **Rotation** from the **Transformation** menu, **Lines** from the **Entities Type** menu and **Two dimensions**. Enter an angle of 120 degrees and select the point (0 25 0), thus rotating the last line created 120 degrees.
3. In the **Copy** window, choose **Translation** from the **Transformation** menu and **Lines** from the **Entities Type** menu. The translation vector for the translation to be made is the line just created. As the first point of the translation, select the point farthest from this line segment. For the second point, select the other point of the line. (Figure 5)

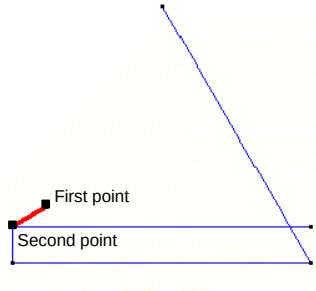


Figure 2. The line segment selected is the translation vector.

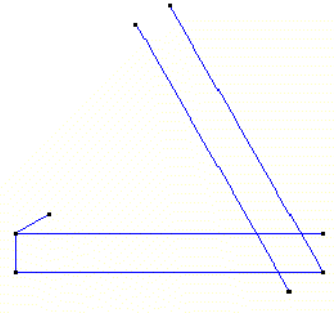


Figure 6. Result of the translation with copy

4. Click **Select** to select the line segment that forms an angle of -60 degrees with the horizontal. Press **ESC** to indicate that the selection has been made.
5. Choose **Geometry→Edit→Intersection→Lines**.
6. Select the two inner lines.
7. The intersection between the two entities (lines) creates a point. This point will be the tangential center.

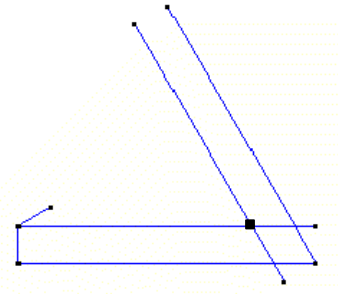


Figure 3. The auxiliary lines

***NOTE**: The option **Undo** enables the user to undo the operations most recently carried out. If an error is made, go to **Utilities→Undo**; a window comes up in which to select all the options to be eliminated.*

3. CREATING THE FIRST COMPONENT PART

In this section the entire model, except the T junction, will be created. The model to be created is composed of two pipes forming a 60-degree angle. To start with, the first pipe will be created. This pipe will then be rotated to create the second pipe.

3.1. Creating the profile

STEP 3

1. Select the **ok** layer by double click. From now on, all entities created will belong to the **ok** layer.
2. Choose the option **Line**, located in **Geometry→ Create→Line.**
3. Enter the following points: (0, 11), (8, 11), (8, 31), (11, 31), (11, 11) and (15, 11). Press **ESC** to indicate that the process of creating lines is finished.

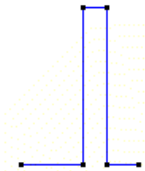


Figure 8. Profile of one of the disks around the pipe

4. From the **Copy** window, choose **Lines** and **Translation**. A translation defined by points (0, 11) and (15, 11) will be made. In the **Multiple copies** option, enter 8 (the number of copies to be added to the original). Select the lines that have just been drawn.

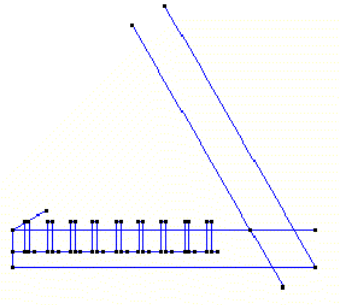


*Figure 4. The profile of the disks using **Multiple copies***

5. Choose **Line**, located in **Geometry→ Create→Straight Line.** Select the last point on the profile using the option **Join C-a**, which is in **Contextual** on the mouse menu. Now choose the option **No join C-a**. Enter point (200, 11). Press **ESC** to finish the process of creating lines.
6. Again, choose the **Line** option and enter points (0, 9) and (200, 9). Press **ESC** to conclude the process of creating lines. (Figure 10)

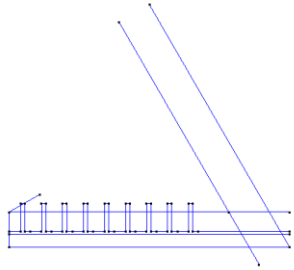


Figure 5. Creating the lines of the profile .

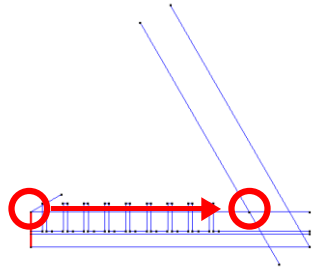


Figure 11. Copy of the vertical line segment starting at the origin of coordinates

7. From the **Copy** window, choose **Lines** and **Translation**. As the first and second point of the translation, enter the points indicated in Figure 11. Click **Select** and select the vertical line segment starting at the origin of coordinates. Press **ESC**.
8. Choose **Geometry→Edit→Intersection→Lines.** Select the two last lines created and the vertical line segment coming down from the tangential center. (See Figure 12.) Press **ESC**.

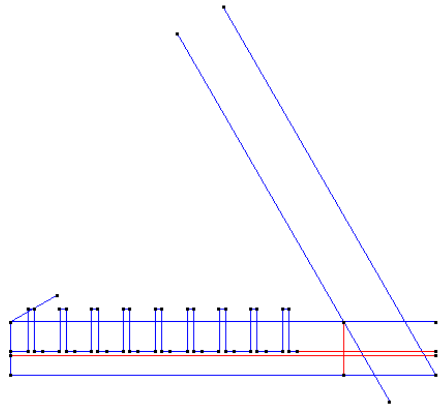


Figure 12. Selecting the lines to intersect

9. Choose **Geometry→Delete→All Types**. (This tool may also be found in the VULCAN Toolbox.) Select (to delete) the lines and points beyond the vertical that passes through the tangential center. Press **ESC**. The result should look like that shown in Figure 13.

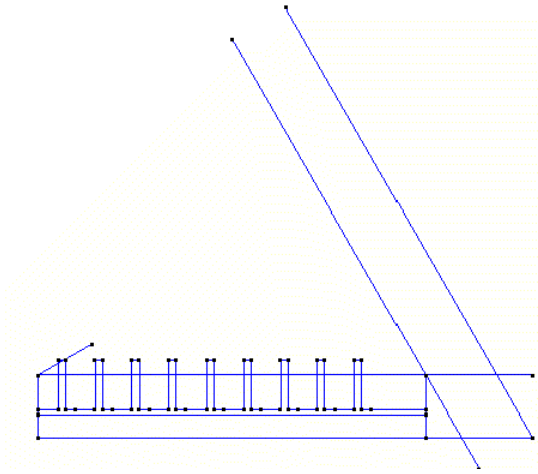


Figure 13. Profile of the pipe and the auxiliary lines

3.2. Creating the volume by revolution

STEP 4

1. Rotation of the profile will be carried out in two rotations of 180 degrees each. This way, the figure will be defined by a greater number of points.
2. From the **Copy** window, select **Lines** and **Rotation**. Enter an angle of 180 degrees and from the **Do extrude** menu, select **Surfaces**. The axis of rotation is that defined by the line that goes from point (0, 0) to point (200, 0). Enter these two points as the **First Point** and **Second Point**. Be sure to enter 1 in **Multiple Copies**.
3. Click **Select.** For an improved view when selecting the profile, click **Off** the "aux" layer. Press **ESC** when the selection is finished. The result should be that illustrated in Figure 14.

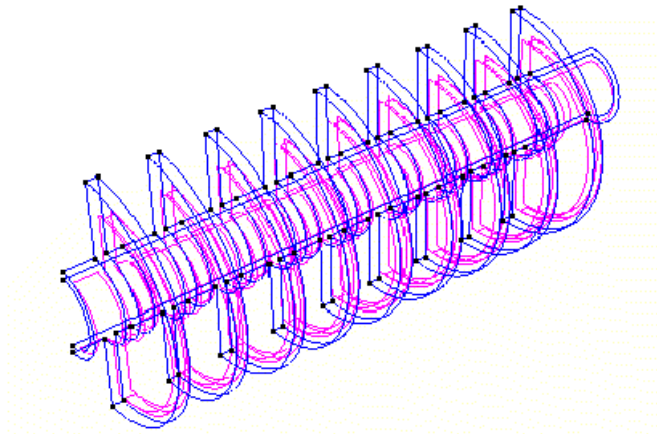


Figure 14. Result of the first step in the rotation (180 degrees)

4. Repeat the process, this time entering an angle of –180 degrees.
5. To return to the side view (elevation), choose **Rotate→Plane XY**.
6. Choose **Render→Flat lighting** from the mouse menu to visualize a more realistic version of the model. Return to the normal visualization with **Render→Normal**. This option is more comfortable to work with.

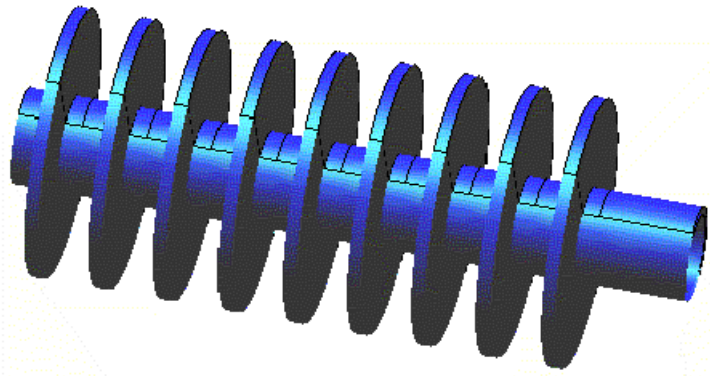


Figure 15. The pipe with disks, created by rotating the profile.

***NOTE**: To select the profile once the first rotation has been done, first select all the lines and then delete those that do not form the profile. Use the option **Rotate→Trackball** from the mouse menu to rotate the model and facilitate the process of selection.*

3.3. Creating the union of the main pipes

STEP 5

1. Choose the option **Zoom→In** from the mouse menu. Magnify the right end of the model.

2. Make sure the "aux" layer is visible.

3. From the **Copy** window, select **Lines** and **Rotation**. Enter an angle of 120 degrees and from the **Do extrude** menu, select **Surfaces**. Since the rotation may be done in 2D, choose the option **Two Dimensions**. The center of the rotation is the tangential center.

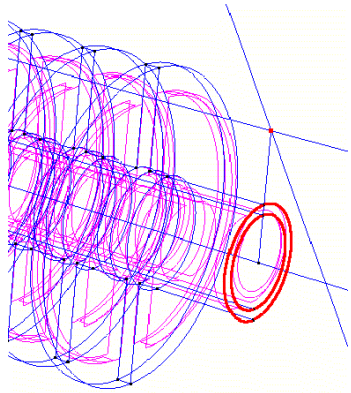


Figure 16. The magnified right end of the model

4. Click **Select** and select the four lines that define the right end of the pipe. (See Figure 16.) Press **ESC** when the selection is finished.

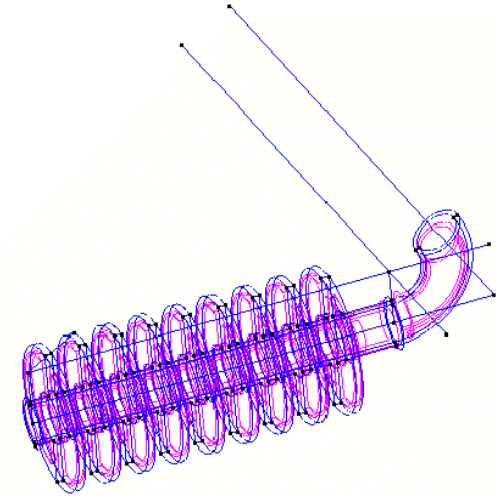


Figure 17. Result of the rotation

3.4. Rotating the main pipe

STEP 6

1. From the **Copy** window, select **Surfaces** and **Rotation**. Enter an angle of -60 degrees. Since the rotation may be done in 2D, choose the option **Two Dimensions**. The center of the rotation is the intersection of the axes, namely, point (200, 0). Be sure the **Do Extrude** menu is in the **No** mode.
2. Click **Select** and select all the surfaces except those defining the elbow of the pipe. Press **ESC** when the selection is finished.

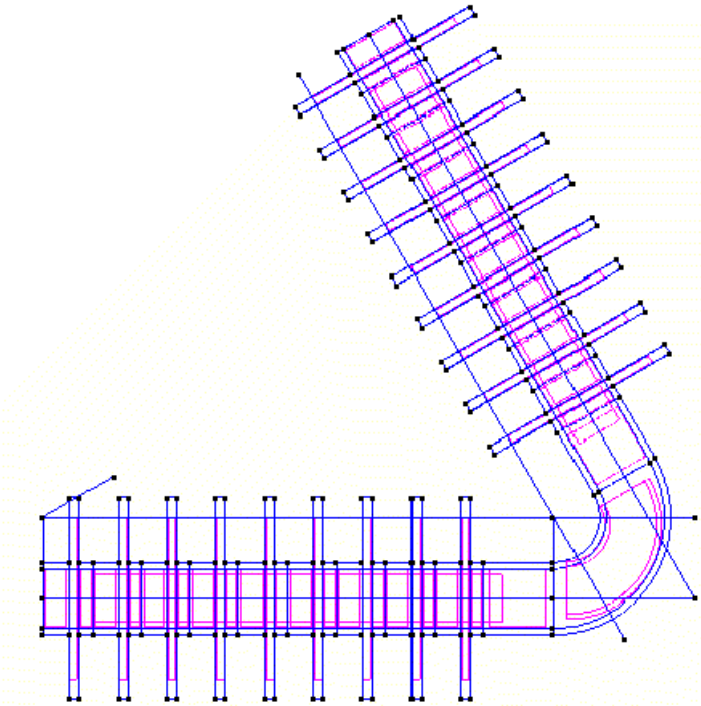


Figure 18. Geometry of the two pipes and the auxiliary lines

3.5. Creating the end of the pipe

STEP 7

1. From the **Copy** window, select **Surfaces** and **Rotation**. Enter an angle of 180 degrees. Since the rotation may be done in 2D, choose the option **Two Dimensions**. The center of rotation is the upper right point of the pipe elbow. Make sure the **Do Extrude** menu is in the **No** mode.
2. Click **Select** and select the surfaces that join the two pipe sections.
3. In the **Move** window, select **Surfaces** and **Translation**. The points defining the translation vector are circled in Figure 19.
4. *Click **Select** and select the surfaces to be moved. Press **ESC.***

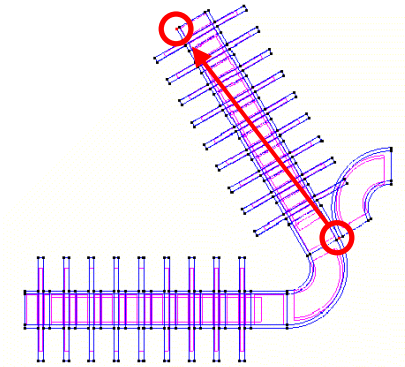


Figure 19. The circled points define the translation vector.

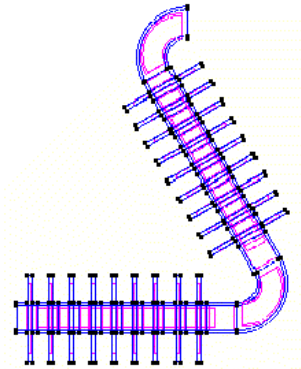


Figure 20. The final position of the translated elbow.

5. Choose **Geometry→ Create→ NURBS Surface→By contour** and select the four lines that define the opening of the pipe. (Figure 21) Press **ESC**.
6. Go to **Geometry →Edit →Collapse →Model** in order to collapse the isolated lines.
7. From the **Files** menu, choose **Save** in order to save the file. Enter a name for the file and click **Save**.

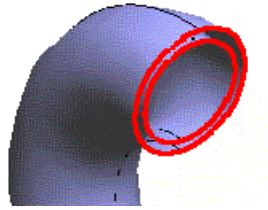


Figure 21. Opening at the end of the pipe

4. CREATING THE SECOND COMPONENT PART: THE T JUNCTION

Now, an intersection composed of two pipe sections will be created in a separate file and the surfaces will be trimmed. Then this file will be imported to the original model to create the entire part.

4.1. Creating one of the pipe sections

STEP 8

1. Choose **Files→New**, thus starting work in a new file.
2. Choose **Geometry→Create→Point** and enter points (0, 9) and (0, 11). Press **ESC** to conclude the creation of points.
3. From the **Copy** window, select **Points** and **Rotation**. Enter an angle of 180 degrees and from the **Do extrude** menu, select **Lines**. The axis of rotation is the x axis. Enter two points defining the axis, one in **First Point** and the other in **Second Point**, for example, (0, 0, 0) and (100, 0, 0). (Figure 22)
4. Click **Select** and select the two points just created.
5. Repeat the process, this time entering an angle of –180 degrees, thus creating the profile of the pipe section with a second rotation of 180 degrees. The rotation could have been carried out in only one rotation of 360 degrees. However, in the present example, each circumference must be defined between two points. (Figure 23)

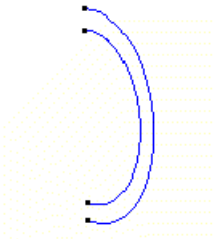


Figure 22. The result of the first 180-degree rotation

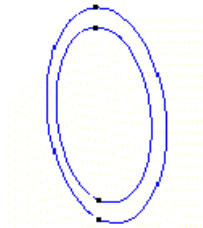


Figure 23. The combined result of the first rotation and the second rotation of –180 degrees, thus obtaining the profile of the pipe section

6. From the **Copy** window, choose **Lines** and **Translation**. In **First Point** and **Second Point**, enter the points defining the translation vector. Since the pipe section must measure 40 length units, the vector is defined by points (0, 0, 0) and (-40, 0, 0).
7. From the **Do extrude** menu, choose the option **Surfaces.**

8. Click **Select** to select the lines that define the cross-section of the pipe. Press **ESC** to conclude the selection process.

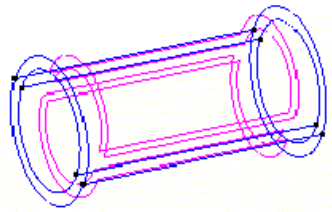


Figure 24. Creating a pipe by extruding circumferences

4.2. Creating the other pipe section

STEP 9

1. Choose **Geometry→Create→Point** and enter points (-20, 9) and (-20, 11). Press **ESC** to conclude the creation of points.
2. From the **Copy** window, select **Points** and **Rotation**. Enter an angle of 180 degrees and from the **Do extrude** menu, select **Lines**. Since the rotation may be done on the **xy** plane, choose **Two Dimensions**. The center of rotation is (-20, 0, 0).
3. Click **Select** and select the two points just created. Repeat the process, this time entering an angle of -180 degrees.
4. From the **Copy** window, select **Lines** and **Translation**. In **First Point** and **Second Point,** enter the points defining the translation vector. Since this pipe section must also measure 40 length units, the vector is defined by points (0, 0, 0) and (0, 0, 40).
5. From the **Do extrude** menu, select the option **Surfaces.**
6. Click **Select** to select the lines that define the cross-section of the second pipe. Press **ESC** to conclude the selection.

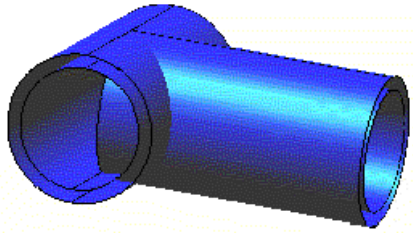


Figure 25. A rendering of the two intersecting pipes

4.3. Creating the lines of intersection

STEP 10

1. Choose **Geometry → Edit → Intersection → Surface-surface**.
2. Select the outer surfaces of each pipe, thus forming the intersection of the two surfaces selected.
3. Repeat the process to obtain the four lines of intersection.

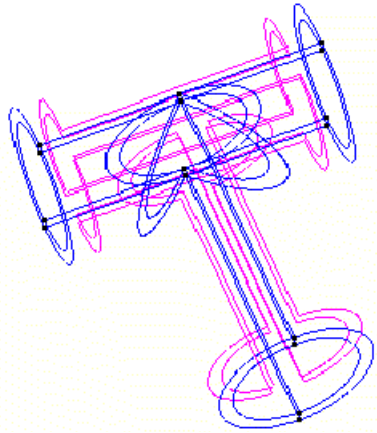


Figure 26. Creating lines of intersection between the surfaces

4.6. Deleting surfaces and lines

STEP 11

1. Choose **Geometry→Delete→Surface**. Select the interior surfaces(Fig 27). Press **ESC** to conclude the process of selection.
2. Choose **Geometry→Delete→Line**. Select the lines defining the end of the second pipe (foreground) that are still inside the first pipe (background).

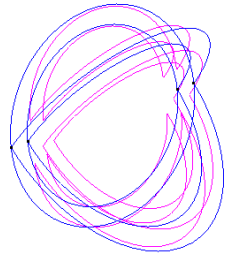


Figure 27. Surfaces to be deleted.

STEP 12

4.7. Closing the volume

1. The model now has three outlets. The two outlets farthest from the origin of coordinates must be closed. The third will be connected to the rest of the piece when the T junction is imported.
2. Choose **Geometry→Create→NURBS Surface→By contour** and select the two lines defining the outlet in the foreground of Figure 30. Press **ESC**. (See Figure 28)

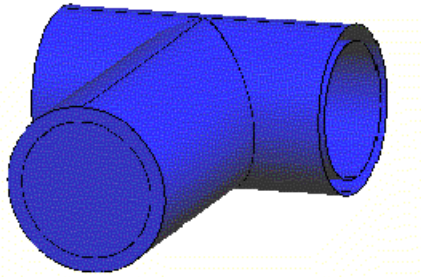


Figure 28. Creating a NURBS Surface to close the outlet in the foreground

3. Choose **Geometry→Edit→Hole NURBS surface**. Select the NURBS surface just created. Then select the lines defining the hole. Press **ESC**. The result is shown in Figure 31.
4. Repeat the process for the other outlet to be closed.
5. From the **Files** menu, select **Save** in order to save the file. Enter a name for the file and click **Save**.

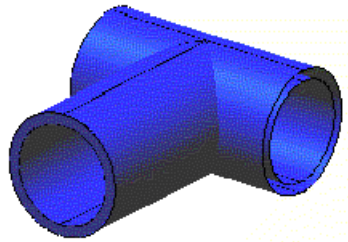


Figure 296. A rendering of the T junction

5. IMPORTING THE T JUNCTION TO THE MAIN FILE

The two parts of the model have been drawn. Now they must be joined so that the final volume may be generated and the mesh generation may be carried out.

5.1. Importing a VULCAN file

STEP 13

1. Choose **Open** from the **Files** menu. Select the file that where the first part, created in section 3, was saved. Click **Open**.
2. Select the "ok" layer, as a **Layer To use** so that the imported file will be in this layer.
3. Choose **Files→Import→Insert VULCAN geometry** from the **Files** menu. Select the file where the second part, created in section 4, was saved. Click **Open**.
4. The T junction appears. Keep in mind that the lines defining the end of the first pipe (background) of the T junction, and which have been imported, were already present in the first file. Notice that the lines overlap. This overlapping will be remedied by collapsing the lines.

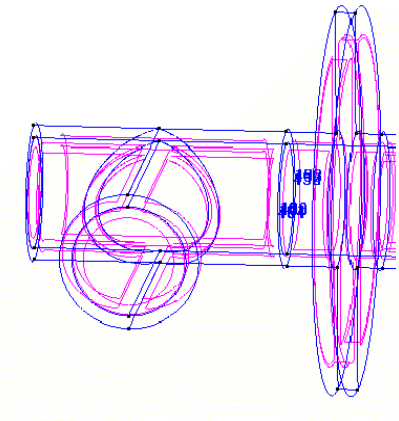


Figure 30. Importing the T junction file to the main file. Some points are duplicated and must be collapsed.

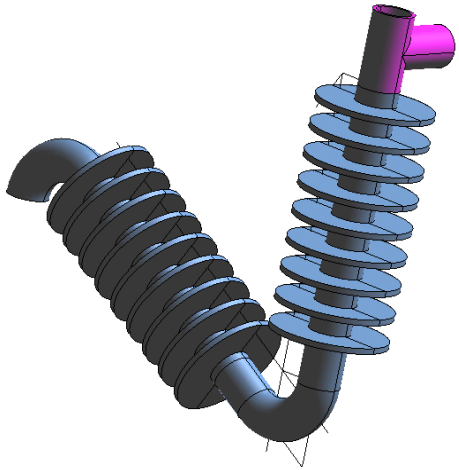


Figure 31. Here you can see that the importation creates a new layer if the names are different.

3. Pass the imported geometry to “ok” layer.
4. Choose the option **Geometry→Edit→Collapse→Lines**. Select the overlapping lines and press **ESC** and delete the sheared surface.

5.2. Creating the final volume

STEP 14

1. Choose **Geometry→Create→Volume** and select all the surfaces defining the volume. Press **ESC** to conclude the selection process.
2. Choose **Render→Smooth lighting** to visualize a more realistic version of the model.

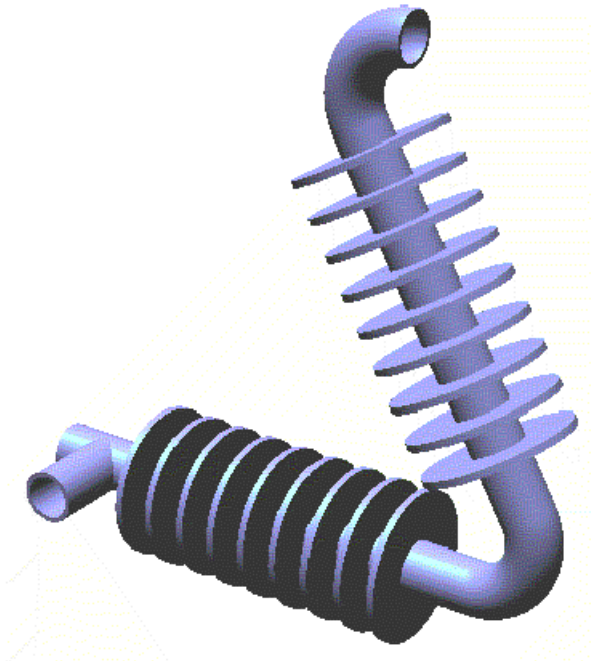


Figure 32. A rendering of the finished piece of equipment

6. GENERATING THE MESH

Now that the model is finished, it is ready to be meshed. The mesh will be generated using **Chordal Error** in order to achieve greater accuracy in the discretization of the geometry. The chordal error is the distance between the elements generated by the meshing process and the real profile of the model. By selecting a sufficiently small chordal error, the elements will be smaller in the zones with greater curvature.

6.1. Generating the mesh using Chordal Error

STEP 15

1. Choose the option **Mesh→Unstructured→Sizes by Chordal error**.
2. The minimum element size is automatically chosen
3. VULCAN asks for the maximum element size. Enter 10 on the command line.
4. VULCAN asks for the chordal error. Enter 0.1
5. Choose **Mesh→Generate Mesh**.
6. A window comes up in which to enter the maximum element size of the mesh to be generated. Assign 8 and click **OK**.
7. When the meshing process is finished, a window appears with information on the mesh that has been generated. Click **OK** to visualize the mesh.
8. Choose **Mesh→View Mesh Boundary** to see only the contour of the volumes meshed but not their interior.
9. This way, the visualization may be rendered using the various options on the **Render** menu, located on the mouse menu.

***NOTE**: By default VULCAN corrects element size depending on the form of the entity to mesh. This correction option may be deactivated or reactivated in the **Preferences** window on the **Meshing** card under the name **Automatic correct sizes**.*

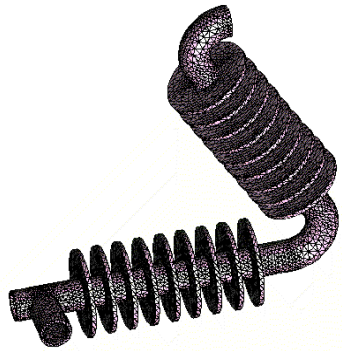
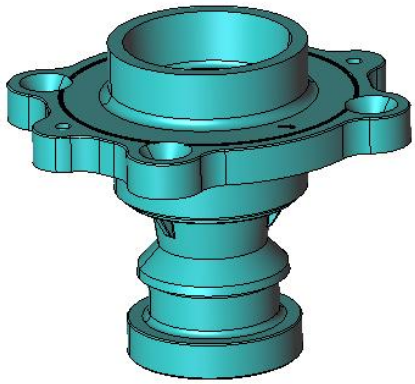


Figure 33. The mesh generated for the pie

D. GEOMETRY PREPARATION

IMPORTING CAD GEOMETRIES INTO VULCAN ENVIRONMENT



Version 10.0

[This page is intentionally left blank]

INTRODUCTION

In Finite Element Analysis, the meshing is a very important step towards the success of the simulation process. Finite element meshes are very precise and they fit very well to the real shape of the casting under study, but sometimes too many details in the part leads to element distortion, or a to a very high number of elements, which will increase the calculation time.

Vulcan is a software that has the tools for geometry preparation, but the dedicated CAD system in which the part has been designed is usually well known by the user, and a few geometry preparation steps previously done in the CAD system will help to export the files and will make the meshing easier; so the preparation should start in the dedicated CAD system, this way will be faster and will drastically reduce preparation in Vulcan.

In this tutorial we will show some procedures that can be done in the CAD system used before exporting the part into Vulcan, and this way avoiding distortions and saving calculation time.

We will see a mesh without corrections, and then a mesh previously corrected before importing into Vulcan, in order to study the differences and understand why it is convenient to simplify some aspects of the geometry under study.

EXPORTING A GEOMETRY DESIGNED IN A CAD ENVIRONMENT

In this tutorial, we are going to generate a Finite Element mesh in Vulcan out of a geometry made in a parametrical CAD environment; we will mesh the following part, generated in CATIA V5 ([1]):

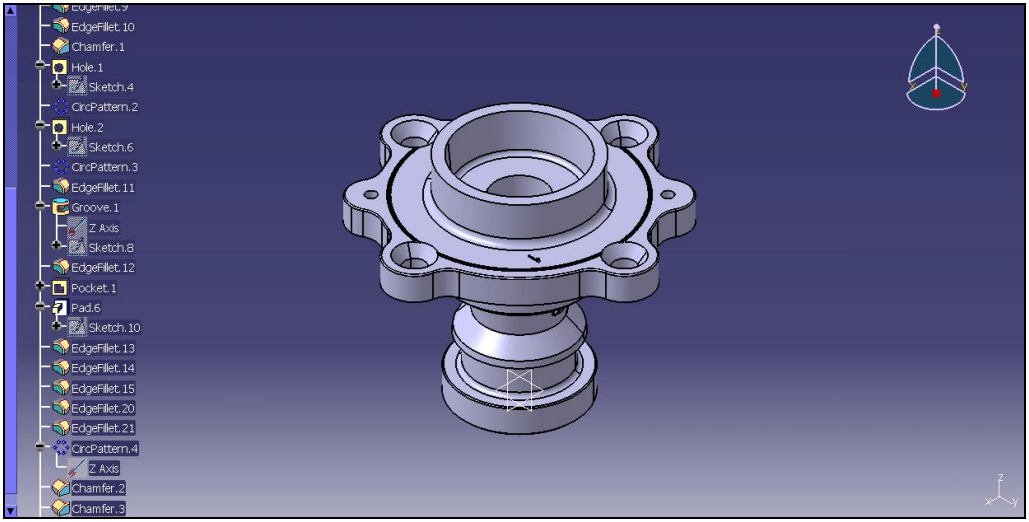


Figure D.1. Part made with CATIA V5 Part Design module.

NOTE ([1]) CATIA and CATIA V5 are registered trademarks of Dassault Systemes.

Now we will import this part into Vulcan interface as an IGES file ([2]):
Go to File → Import → IGES…, and select the file *tutorial_import3.igs*.

This file can be downloaded from the VULCAN web page http://www.quantech.es/support-vulcan.aspx

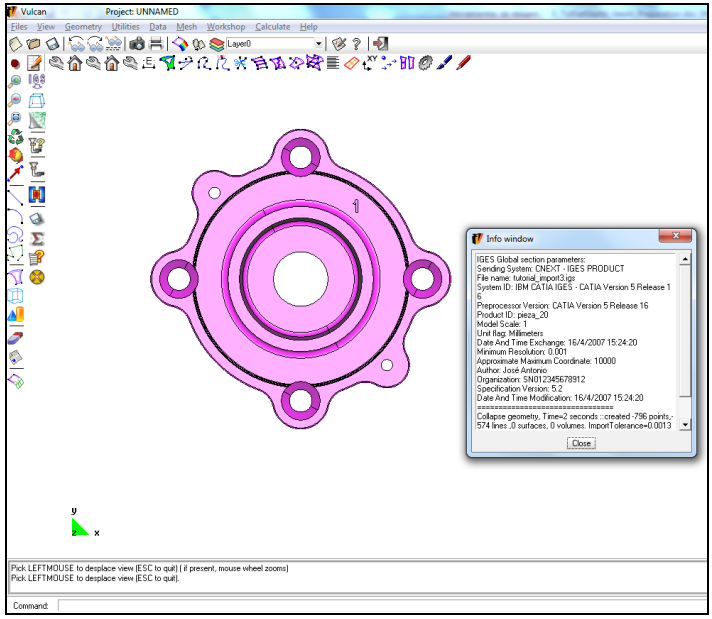

Vulcan
Project: UNNAMED
Files View Geometry Utilities Data Mesh Workshop Calculate Help
Info window
IGES Global section parameters:
Sending System: CNEXT - IGES PRODUCT
File name: tutorial_import3.igs
System ID: IBM CATIA IGES - CATIA Version 5 Release 1
6
Preprocessor Version: CATIA Version 5 Release 16
Product ID: pieza_20
Model Scale: 1
Unit flag: Millimeters
Date And Time Exchange: 16/4/2007 15:24:20
Minimum Resolution: 0.001
Approximate Maximum Coordinate: 10000
Author: José Antonio
Organization: SN01234567891 2
Specification Version: 5.2
Date And Time Modification: 16/4/2007 15:24:20
Collapse geometry. Time=2 seconds : created -796 points,-
574 lines ,0 surfaces, 0 volumes. ImportTolerance=0.0013
Close
Pick LEFTMOUSE to desplace view [ESC to quit] (if present, mouse wheel zooms)
Pick LEFTMOUSE to desplace view [ESC to quit].
Command:

Figure D.2. IGES file importing in Vulcan.

NOTE ([2]) For more details about importing and exporting, see Meshing section in Vulcan user's Manual.

Now we will have a look over the imported part: we see it in render view, we check the higher entities, we see the details, etc. (Note that this part is only a surface geometry):

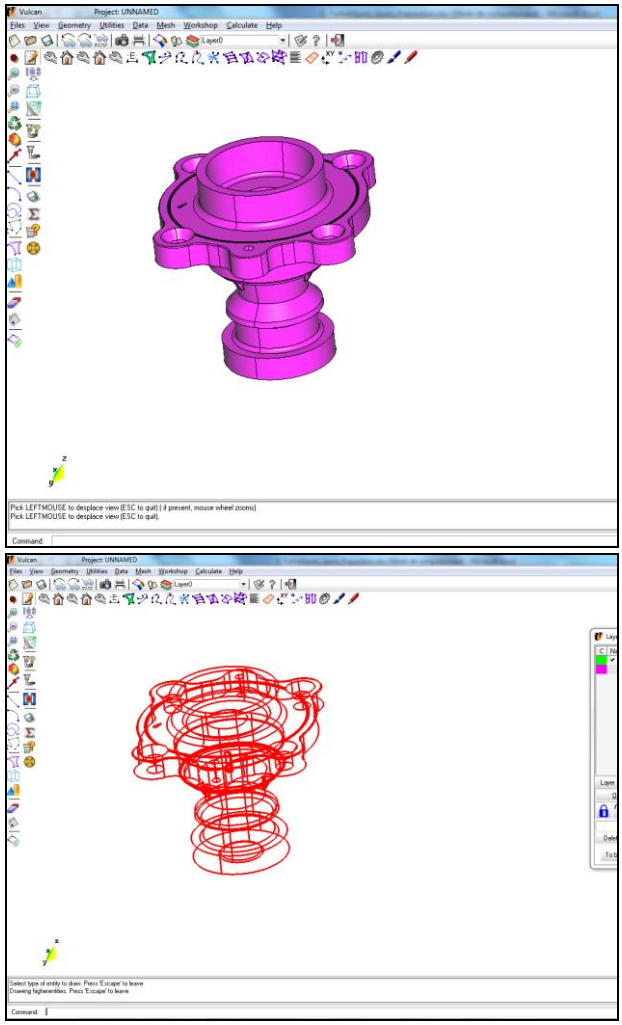


Figure D.3. Renderizing and Higher entities of the imported part.

This part, as we see in Figure D.3, is a set of closed surfaces, as we see when we check with higher entities; all the surface entities are well connected, and there are no volumes generated.

Due to the characteristics of Vulcan mesher, surface mesh will be generated prior to volume mesh; furthermore, volume tetrahedral mesh will be based on the surface triangular mesh and will be generated advancing from this surface mesh; so it is always necessary, for complex parts, to firstly generate the surface mesh in order to check its quality regarding to distortion.

Now, go to: Mesh → generate mesh…

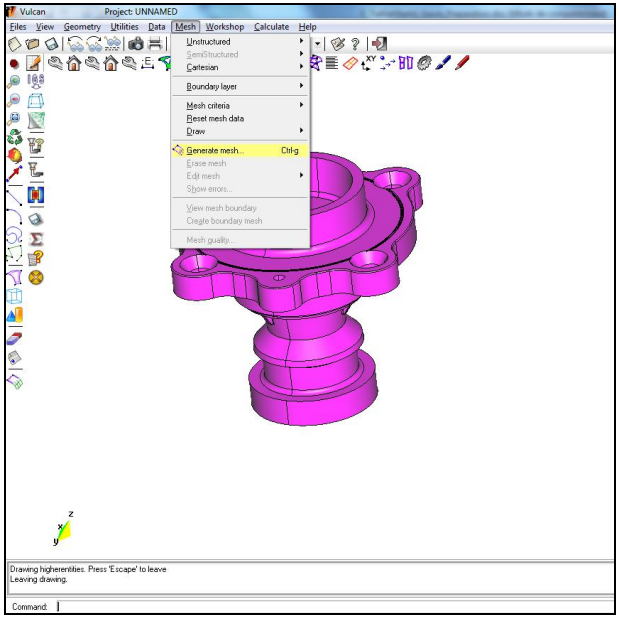


Figure D.4. Mesh generation.

And enter an element size of 3:

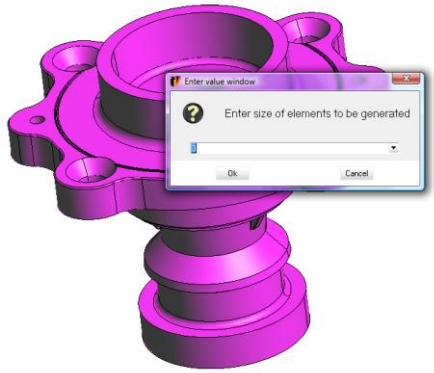


Figure D.5. Mesh size.

This is the result we get:

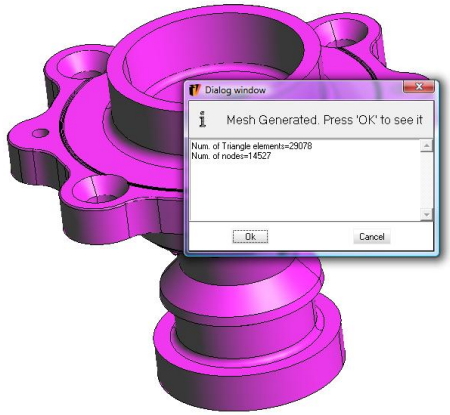


Figure D.6. Generated mesh.

Now after clicking **View mesh** the meshed part will appear (Figure D.7):

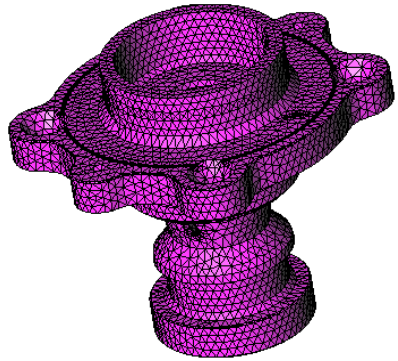


Figure D.7: Triangular Finite Element mesh of the imported part.

The mesh of Figure D.7 is generated without errors, but there are some areas in which the distortion of the elements could generate problems when it comes to calculation, volume meshing, etc. It is important to correct as much as possible all the distorted elements.

Now we will examine the quality of this surface mesh by having a look at them ([3]) and finding zones of distorted elements:

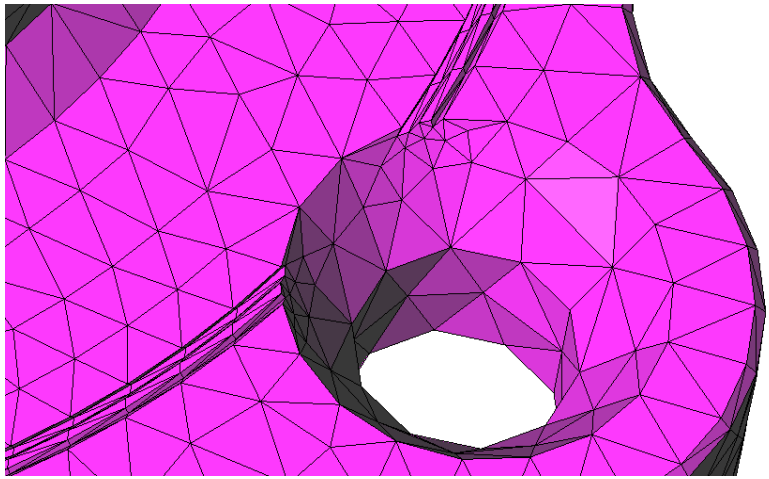


Figure D.8. A machining groove done after the casting is also drawn in the part, causing distorted elements. It is convenient to delete the groove.

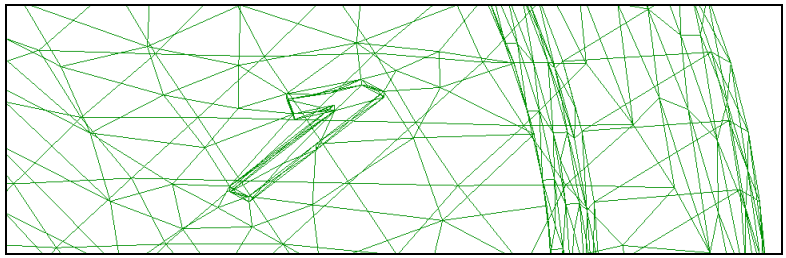


Figure D.9. An engraved text ("Pocket" in CATIA) representing a part number; this text is causing distorted and small elements, and it is convenient to delete it.

NOTE ([3]) There is also a Mesh quality tool that can be used to check the mesh. For more information, see Meshing section on Vulcan manual.

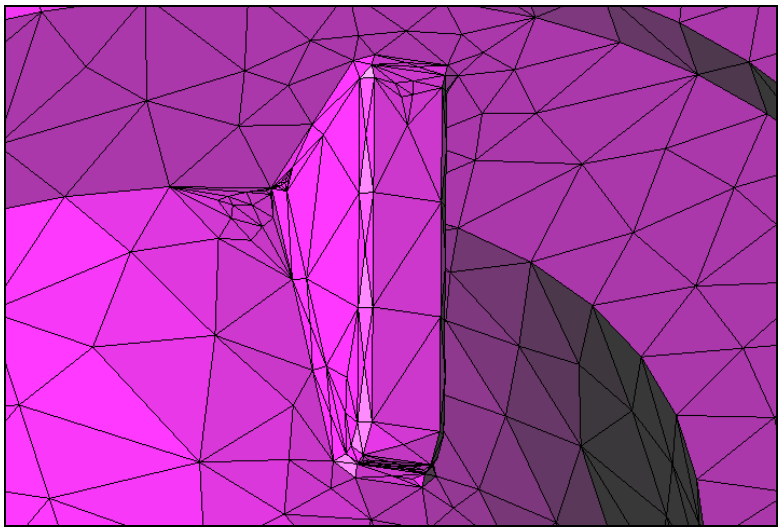


Figure D.10. Element distortion caused by small radii ("edge fillets") in some zones of the part.

Even though all these mesh distortions can be corrected with Vulcan, there are some measures that we can take *before* we export the part into an IGES file.

Machining operations

All the machining operations made after the casting over the part (slots, grooves), can be excluded before we export the part.

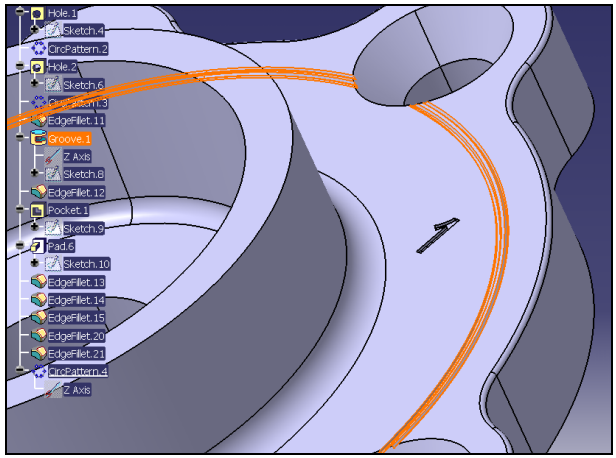
Small radii and small chamfers

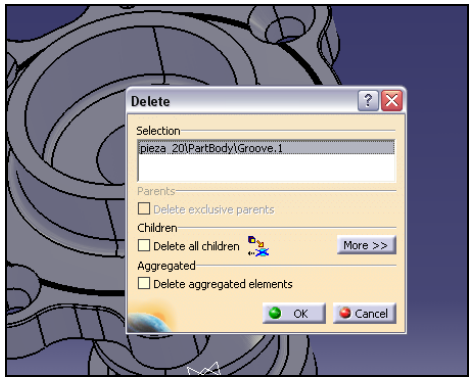
We can also exclude the small fillets and chamfers of the part.

Texts and logos embossed or engraved

They usually don't represent the body of the part and generate distortions.

To do this, we go back to CATIA and have a delete the unneeded operations by using the tree:





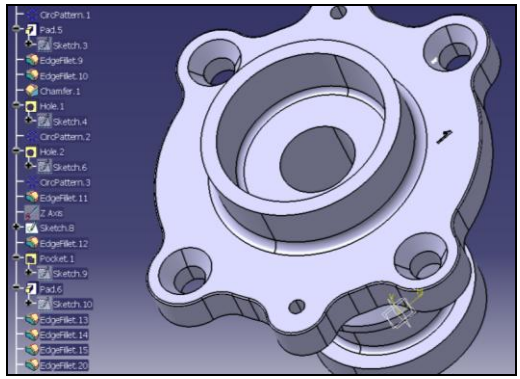


Figure D.11. Groove operation before and after deleting it.

Now we will delete a Pocket operation:

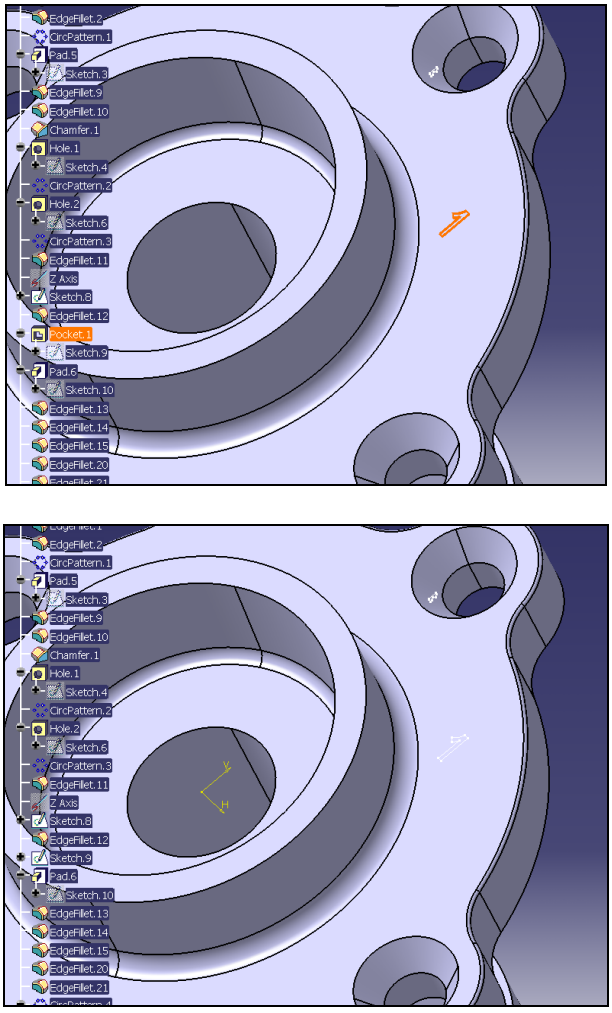


Figure D.12. Pocket operation deletion.

Here wee see EdgeFillet operations:

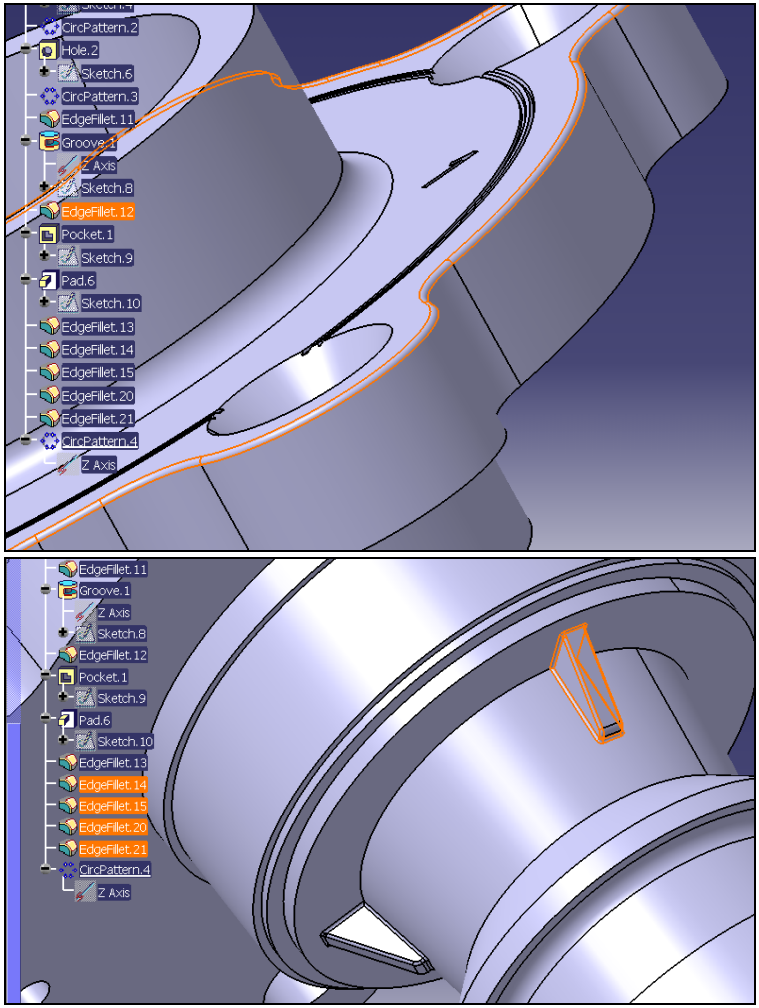


Figure D.13. EdgeFillet operations.

And after the deletion, the same parts will look like this:

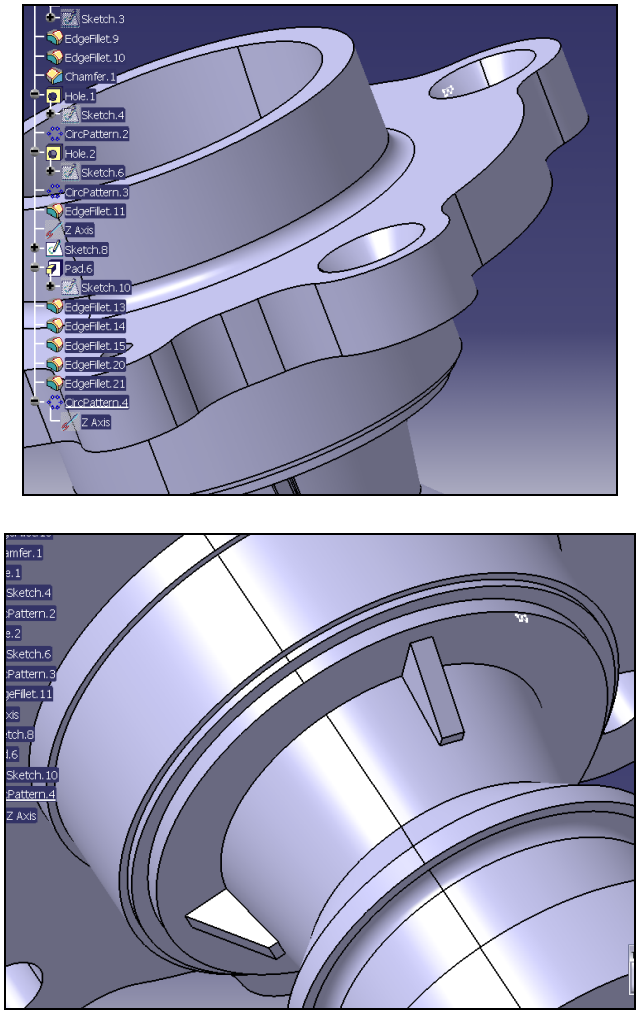


Figure D.14. Edge fillets deleted.

And finally, the deletion of Chamfer operations:

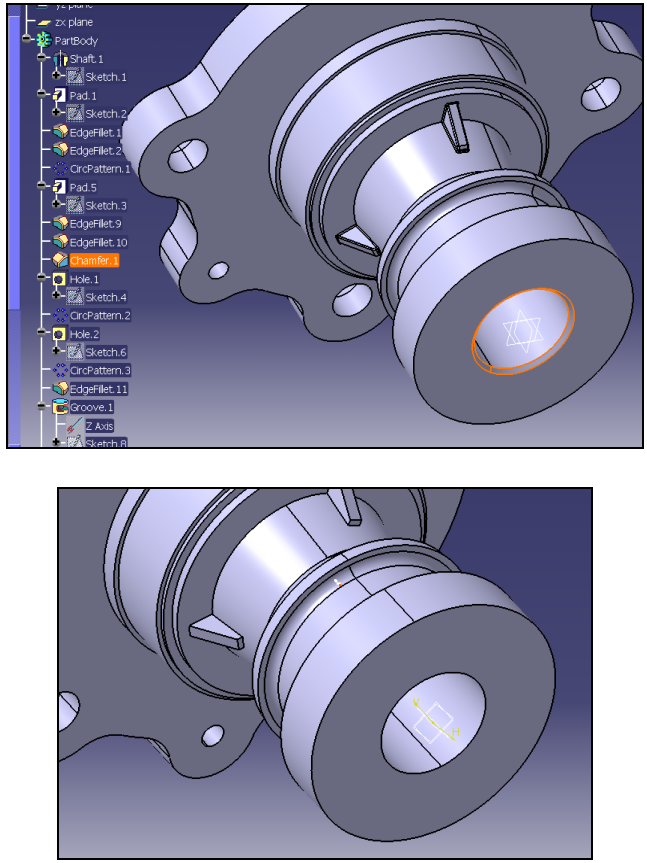


Figure D.15. Chamfer operation deletion.

Once all operations from Figure D.11 to Figure D.15 are deleted, the quality of the mesh generated in Vulcan environment will automatically increase; by working in this manner, we can decrease the meshing time of our part in Vulcan.3

Now, let's see how the part without small radii, etc. will look in Vulcan.

Go to Files → New
Go to Files → Import → IGES…, and select the file *tutorial_import4.igs*.

This file can be downloaded from the VULCAN web page http://www.quantech.es/support-vulcan.aspx

After meshing the part, we can see the result and compare the effect of removing the fillets, machining operations, etc.:

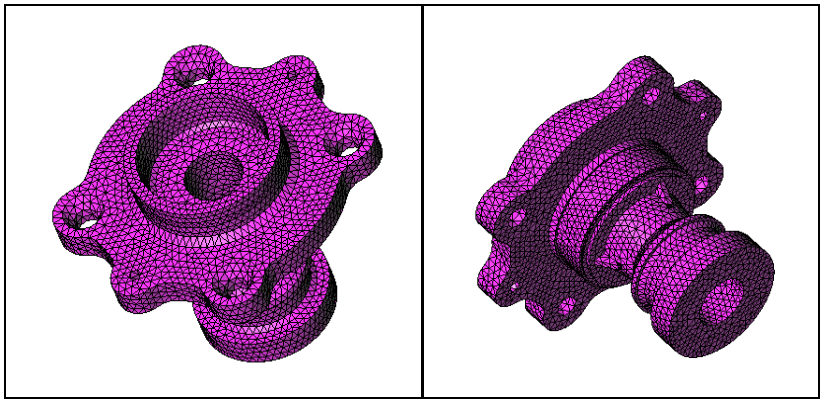


Figure D.16. Part meshed without distortion.

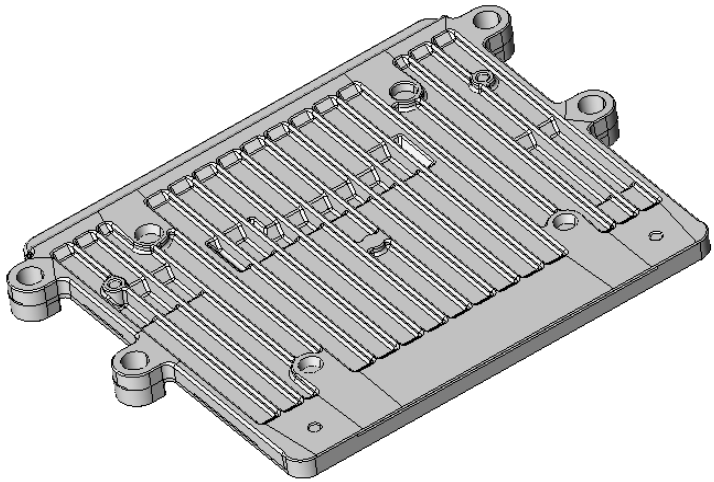
We can compare the results of Figure D.16 with the previous mesh, Figure D.7 to Figure D.10.

E. GEOMETRY CORRECTION

The objective of this tutorial is to see how VULCAN imports files created with any other CAD software. The imported geometry may contain imperfections that must be corrected before generating the mesh.

For this study, an IGES formatted geometry representing a casting is imported. These steps are followed:

- Importing an IGES formatted file to VULCAN
- Correcting errors in the imported geometry
- Generating the correct mesh



[This page is intentionally left blank]

1. IMPORTING AN IGES FILE

VULCAN is designed to import a variety of file formats. Among them are standard formats such as IGES, DXF, Parasolid or VDA, which are generated by most CAD programs. VULCAN can also import meshes generated by other software, in NASTRAN or STL formats.

The file importing process is not always error-free. Sometimes the original file has incompatibilities with the format required by VULCAN. These incompatibilities must be overcome manually. This example deals with various solutions to the difficulties that may arise during the importing process.

1.1. Importing an IGES file

1. Select **Files→Import→IGES…**
2. Select the IGES-formatted file "tutorial_importi5.igs" and click **Open**.

This file can be downloaded from the VULCAN web page http://www.quantech.es/support-vulcan.aspx

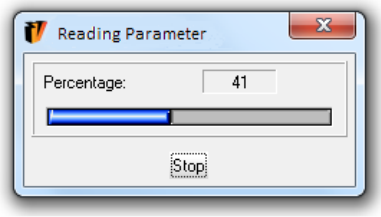


Figure 1. Reading the file.

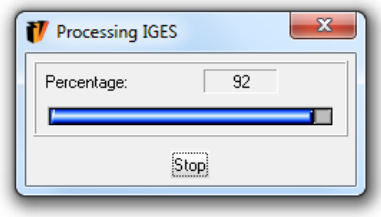


Figure 2. Translating the IGES format.

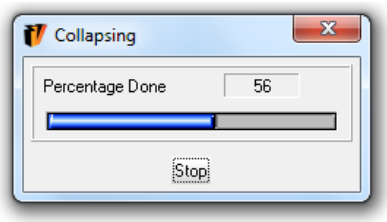


Figure 3. Collapsing the model.

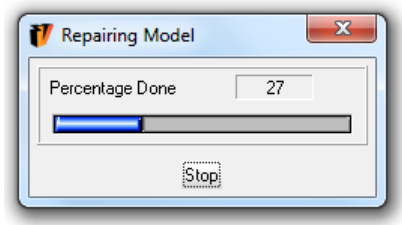


Figure 4 Repairing model.

Figure E.1. IGES importing progress.

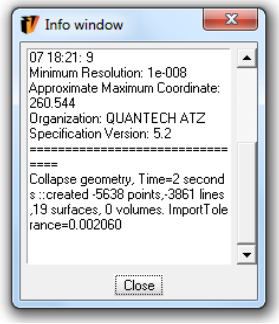


Figure E.2. Importing process information

3. After the importing process, the IGES file that VULCAN has imported appears on the screen.

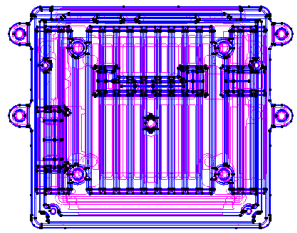


Figure E.3. File "tutorial_importi5.igs" imported by VULCAN.

Select ***Right button→Render→Flat***

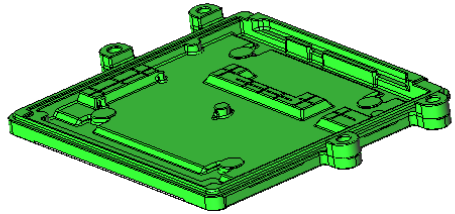


Figure E.4. Renderization of the geometry in which we can see pieces lying on different layers.

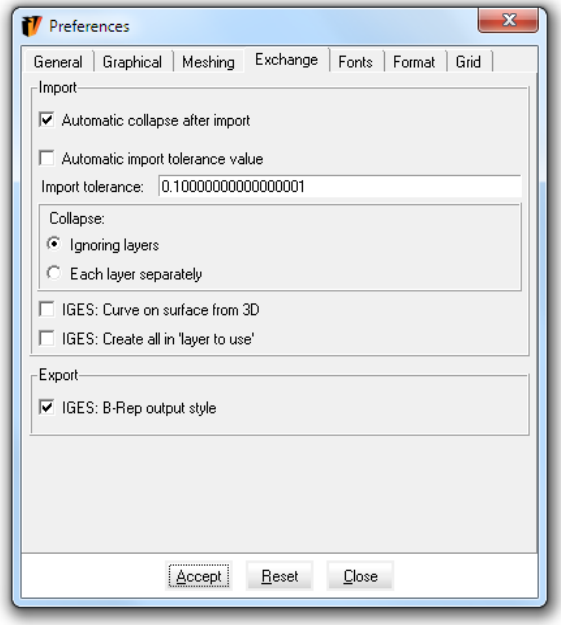


Figure 5. The **Preferences** window.

4. Go to the **automatic import tolerance value** and change the default value to **0.1** in order to collapse some lines that we cannot collapse with the previous value. And now we can see with the **higher entities** how we have reduced the number of bad lines.

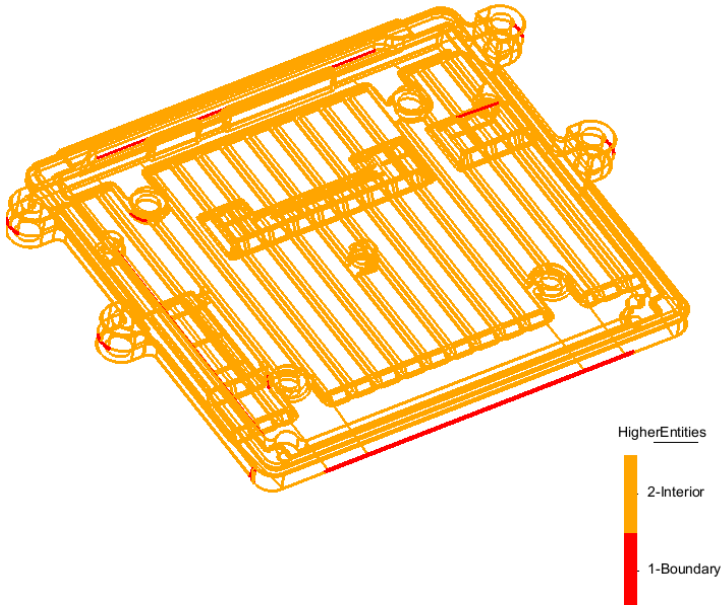
***NOTE**: One of the operations in the importing process is collapsing the model (Figure 3). We say that two entities collapse when, being separated by a distance less than the so-called **Import Tolerance**, they become one. The **Import Tolerance** value may be modified by going to the **Utilities** menu, opening **Preferences**, and bringing up the Import card. By default, the **Automatic import tolerance value** is selected. With this option selected, VULCAN computes an appropriate value for the **Import Tolerance** based on the size of the geometry. Collapsing the model may also be done manually. This option is found in Utilities → **Collapse → Model**.*

2. Correcting ERRORS IN THE IMPORTED GEOMETRY

The great diversity of versions, formats, and software frequently results in differences (errors) between the original and the imported geometry. With VULCAN these differences might result into imperfect meshes or prevent meshing altogether. In this section we will see how to detect errors in imported geometry and how to correct them.

2.1. Correcting the geometry

1. If we select the higher entities command, we will see how there are different places with different kinds of errors in this part. If not select **delete lines** and delete all the lost lines of the part (selecting all the part you only will delete the lost lines, not the lines that belongs to a surface)



2. First of all we go to close the holes of the part (5 holes).
   Go to Geomtry→ Create→ Hole Nurbs Surface → By Contour and select the lines that border on the surface.

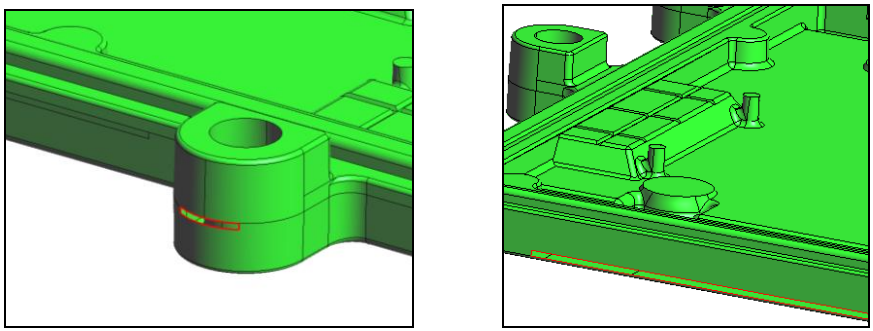


Figure 87. Creation of surfaces.

3. Also we have to close and repair this bad surfaces in one corner of the part. First of all we have to generate the missing surfaces, then with the option join we merge the bad surfaces and finally we collapse all the points of this area.

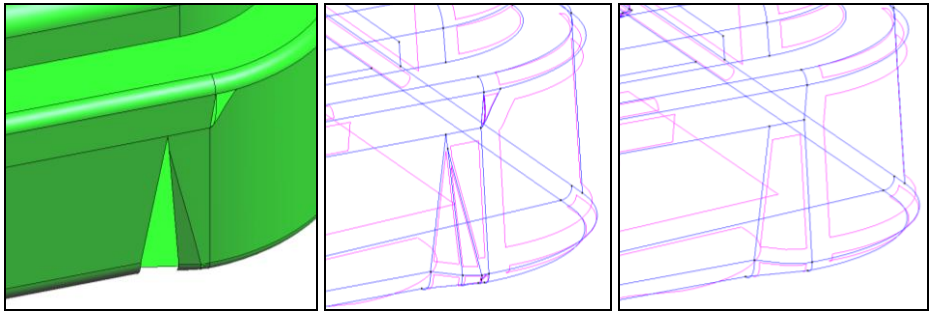


Figure 88. Correction of the corner.

4. Now we have the surface almost ready to generate the volume but first we have to correct these previous issues.
   We can solve these kinds of problems collapsing the lines with higher entities 1.
   But for do this we have to increase the tolerance value.

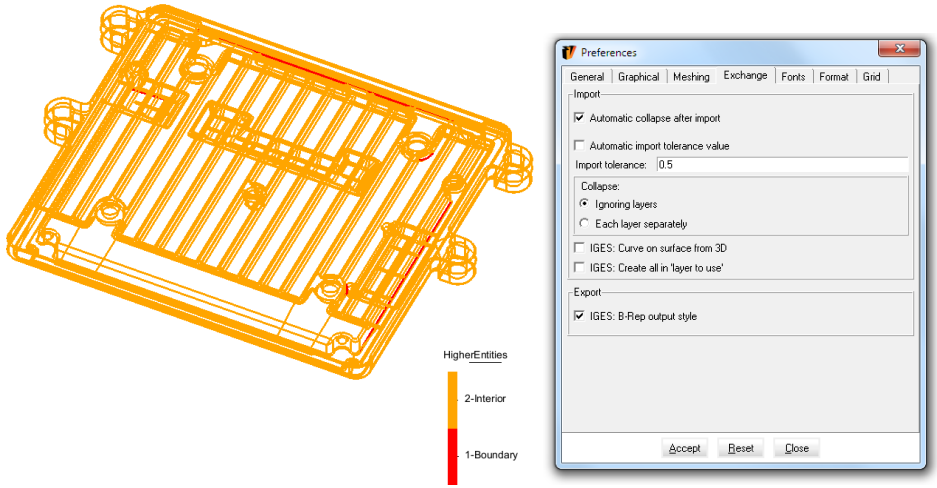


Figure 89. Changing the collapsing tolerance value.

5. We have to be careful to collapse with this value, and only choose the lines that we want to collapse because we can break the part.

6. And we have the part closed, ready to create the volume and mesh.

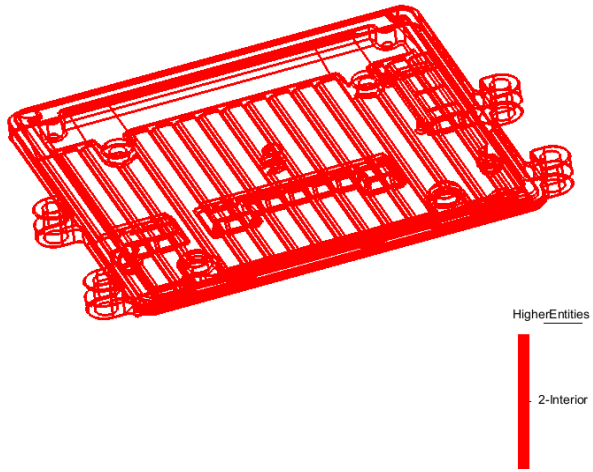
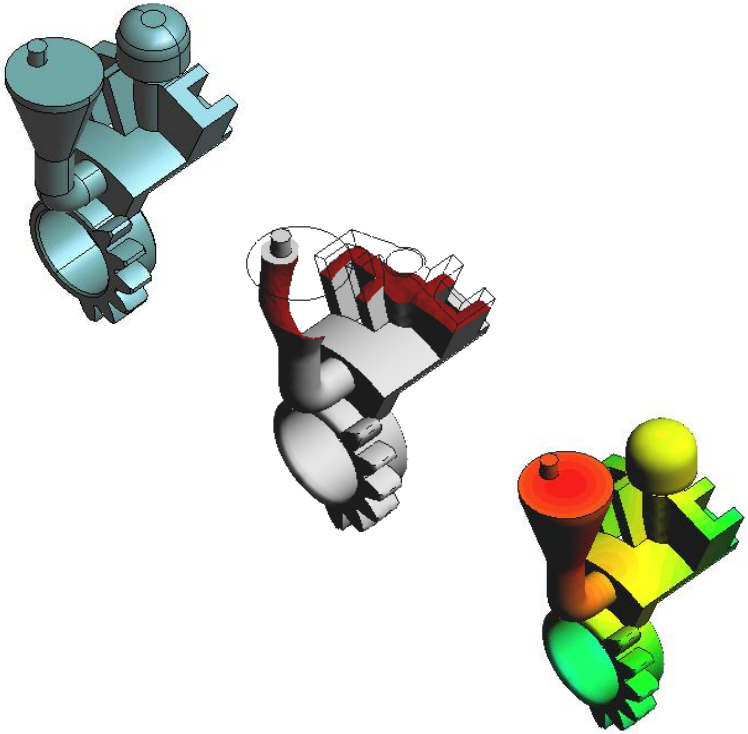


Figure 90. Final part.

F. GRAVITY

GRAVITY CASTING SIMULATION USING VULCAN



Version 10.0

[This page is intentionally left blank]

INTRODUCTION

In this tutorial, we will follow an entire step-by-step procedure in order to run a **gravity casting simulation** ([1]) using Finite Element Analysis with Vulcan.

We will import an **IGES** geometry into Vulcan's graphical interface, mesh it, set the process parameters, and finally get simulation results (temperatures, velocities, turbulences, porosity, defects, etc.)

Basically, a **Finite Element Analysis** consists of three phases:

- Pre-process
- Calculation
- Post-Process

During the **pre-process** phase, we will completely define the problem geometry, set the types of analyses to perform, and establish the physical properties of the materials, temperatures, etc.

Once the problem is completely defined, we will launch the **calculation**; the total calculation time will depend on the complexity of the problem (this is, the number of **elements** in both the casting and the mould). During the calculation time, user's participation is not required. The calculation will let us know when is finished.

When calculation is finished, the user can start the **post-processing** phase; this is the results loading, view and analysis; this is the objective of the finite element simulation.

This tutorial consists of 9 steps, that will guide you trough all the gravity casting simulation process.

Before we start, let's describe Vulcan the interface. You can skip the Step 0 if you have previously worked with the interface.

NOTE ([1])*: The term "Gravity casting simulation" includes* ***gravity sand casting****,* ***gravity die casting*** *and* ***tilt pouring*** *processes.*

Step 0

THE PROCESS BAR

Vulcan has incorporated an entirely new process bar ([2]), that will guide us through all the simulations process of filling, thermal.. of our casting process. This process bar is normally located on the left side of the screen:

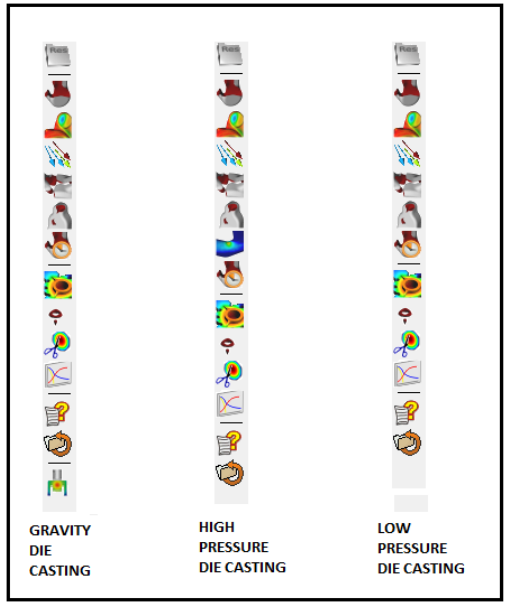


Figure F.1. The process bar.

NOTE ([2])*: If not present, you can activate the Process bar from the menu Utilities → Tools →Toolbars and also change its position to top, bottom, right, etc.*

Process bar icons

1. Load results
2. Visualize filling mater result
3. Visualize filling temperature evolution result
4. Visualize filling vectors result
5. Visualize front encounter result
6. Visualize last air result
7. Visualize mold erosion result
8. Visualize filling time result
9. Visualize thermal temperature evolution result
10. Visualize solid fraction result
11. Cut with a plane the temperature result
12. Visualize the temperature evolution with graphs
13. View process info
14. Reload the results
15. Calculate rise from modulus

These icons are placed in a logical order:

1 Load results
2 to 8 Filling results

9,10,11,12,15	Thermal results
13,14	Process info

Let's begin by opening Vulcan and then with the step-by-step guided tutorial.
On the computer Desktop, double-click on the **Vulcan** icon:

the main screen will show up:

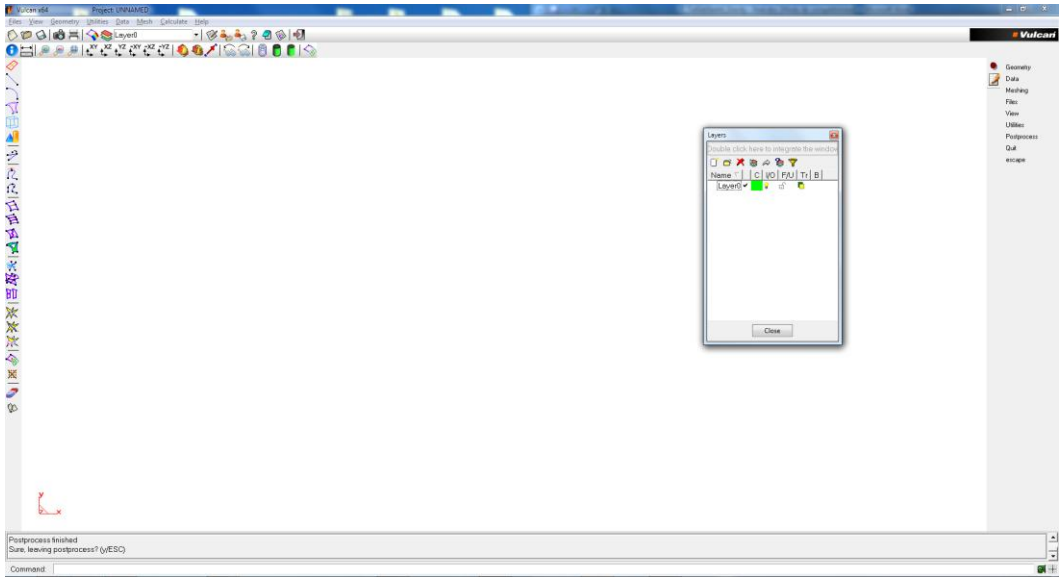


Figure F.2. Vulcan main screen.

Once Vulcan is opened, go to the main menu and click on:
Data → Problemtype → Vulcan

The following window will appear (***Figure F.3***):
(If the window doesn't appear, you can go directly to Step 1)

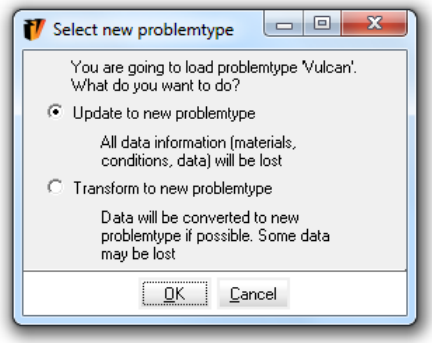

Select new problemtype
You are going to load problemtype 'Vulcan'.
What do you want to do?
Update to new problemtype
All data information (materials, conditions, data) will be lost
Transform to new problemtype
Data will be converted to new problemtype if possible. Some data may be lost
OK
Cancel

Figure F.3. Vulcan problemtype window.

Click OK and the process bar will open automatically; it will be used and described in detail in the present tutorial.

Now that we have the Vulcan pre-process interface opened and set, let's begin with the first step.

Step 1

READ AN IGES FILE

Before reading the IGES file, we are going to set Vulcan parameters in the optimal configuration for importing this type of files. In the main menu, click on Utilities → Preferences…
The following window will open:

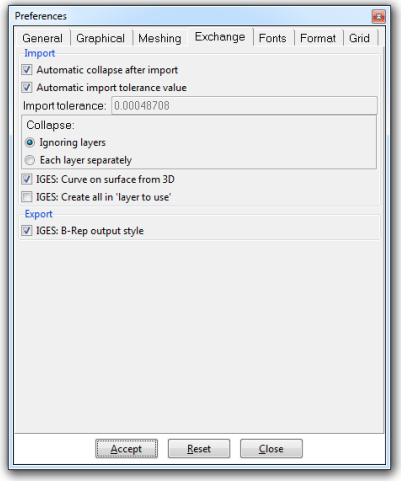


Figure F.4. Setting importing preferences.

The following checkboxes must be selected in ***Figure F.4***:

- ☑ Automatic collapse after import
- ☑ Automatic import tolerance value

Accept the changes and **Close** the window.

Now we are ready to import IGES files. Let's import a sample file.
This file can be downloaded from the VULCAN web page http://www.quantech.es/support-vulcan.aspx

Click the first icon on the process bar:

and read the following file: *Rev_gravity_V8.igs* Select it and click **Open**.
the result should look like this:

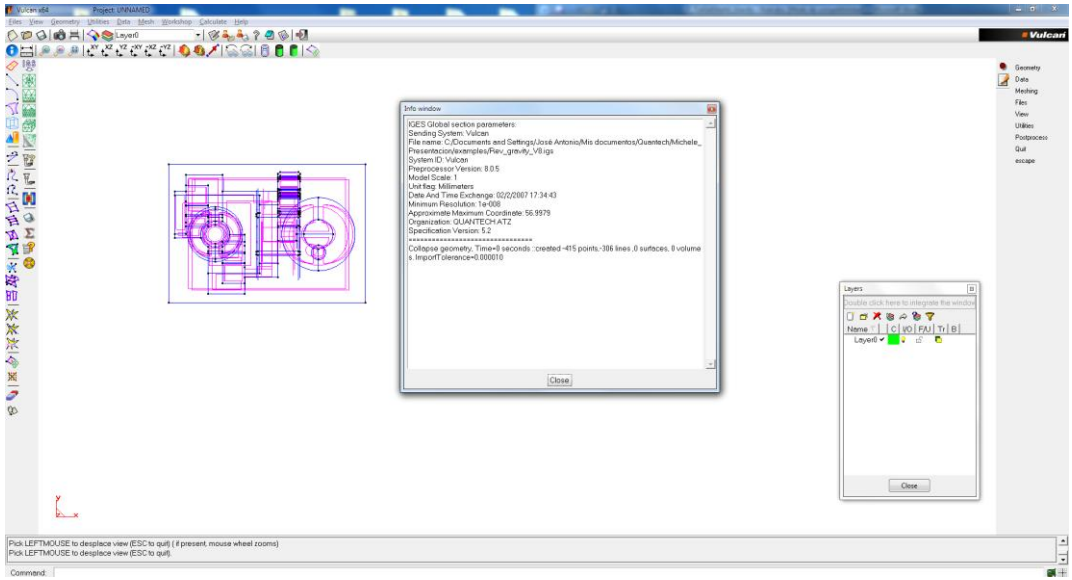


Figure F.5. IGES read.

Close the info window and have a look at the geometry of ***Figure F.5*5** by right-clicking and doing **rotate-trackball** and **zoom in** and **out**.

Once done with looking at geometry, we will start working with different layers: one layer for the casting, one layer for the mould, etc. Click on the Layers icon:

The following window will appear (***Figure F.66***):

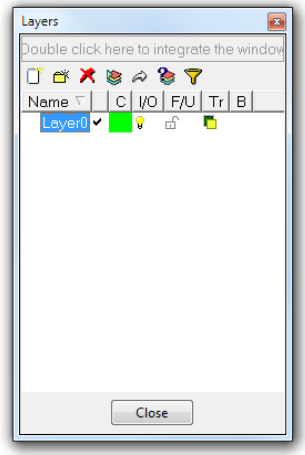


Figure F.6. Layers control window.

Perform the following operations:

- Click on **New** button, and create a new layer.

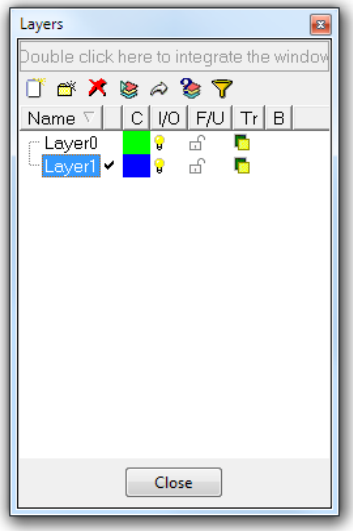


Figure F.7. Layer selection and layer to use.([3])

- **Select** the layer Layer0, change the name “cast” selecting the **Rename** button.
- **Select** the layer Layer1, change the name “mould” selecting the **Rename** button.

NOTE ([3]): The **selected** layer is highlighted in black. After selecting a layer by simply clicking on it, we can delete it, rename it, change its color, turn it on, and off, send elements to it, etc. The **layer to use**, instead, is the layer with the checkmark: in the layer to use we can create geometry, delete it, etc. in the drawing. Summary: **Selected** is to make changes by using layers window, and **layer to use** is the layer used to create, delete, or modify geometry. We can change **the layer to use** by doubleclick.

- Be sure that **Layer to use** is cast; it says so beside the button, and there is a checkmark (✓) in the layer name.
- Select the layer mould, and click on **Send To**, and then Surfaces.
- Go to the casting geometry and select the six surfaces that form the exterior of the mould.
- Turn **off** the layer mould.
- Select the layer cast, and Send **To again**, select all the cast surfaces and press **Esc**
- **Close** the layers window.

The resulting geometry at this moment shall look like ***Figure F.***.

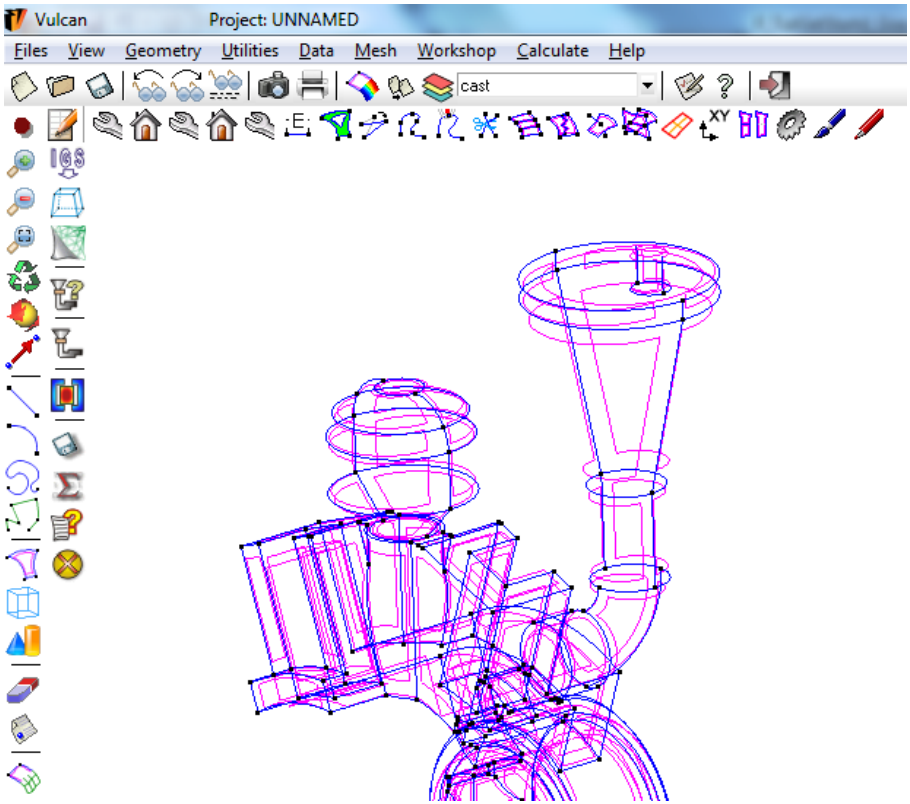


Figure F.8 Gravity casting geometry.

The geometrical entities in Vulcan have the following hierarchy order:

Points → Lines → Surfaces → Volumes

We can interpret the hierarchy by higher entities and lower entities. For example, a surface is higher than a line, a surface is lower than a volume, etc; we cannot delete a surface if there are related higher entities present (Volumes).

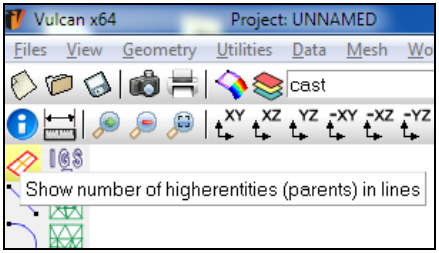
In ***Figure F.8*** we can see the points in black, lines in blue and surfaces in pink. There are always points that define line ends, lines that define surfaces, etc.

Now that we have a geometry to work with, let's begin with the next step, geometry treatment.

Step 2

GEOMETRY TREATMENT

This geometry apparently has no errors, so we won't collapse points or lines. Let's check the integrity of the geometry by selecting **Draw higher** option.


Vulcan x64
Project: UNNAMED
Files View Geometry Utilities Data Mesh Wo
cast
XY XZ YZ -XY -XZ -YZ
Show number of higherentities (parents) in lines

The result shall look like this:

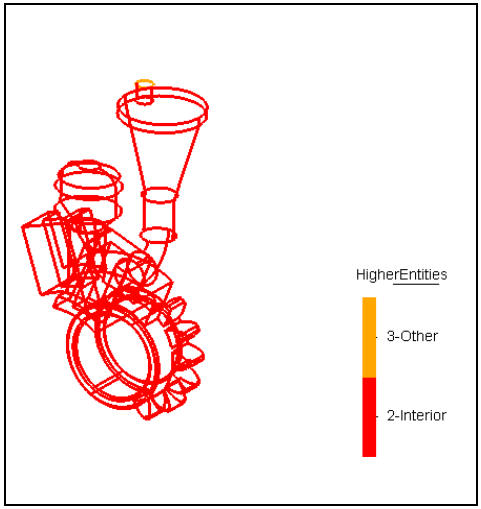


Figure F.9. Draw Higher screen.

In ***Figure F.*** we can check the geometrical integrity of the model before we mesh it. To form a closed surface, all the lines need to be the end of two surfaces ("Interior" in the figure). If there would be open edges on the geometry, the lines would appear under a different colour and labelled "Boundary", or "Isolated" lines.

The lines at the entrance of the material ("Other") are connected to more than two surfaces because they are also connected to the mould surfaces, so they have a different colour; this is correct. The other lines are all "interior" (in red). This means that all the lines are connected to two surfaces (the lines are not forming open boundaries or are isolated lines). In other words, this geometry, consisting of points, lines and surfaces, is sound.

We can rotate, and zoom the casting while we are in **draw higher** mode.

Now let's press **Esc** to go back to the geometry.

We have now a repaired geometry. Let's move on to the next step.

Step 3

MESH GENERATION

Before start the meshing generation, we will set the Meshing parameters as shown in ***Figure F.5***.Go to **Utilities** → **Preferences** and select the label **Meshing**. Set all the parameters as shown in the figure:

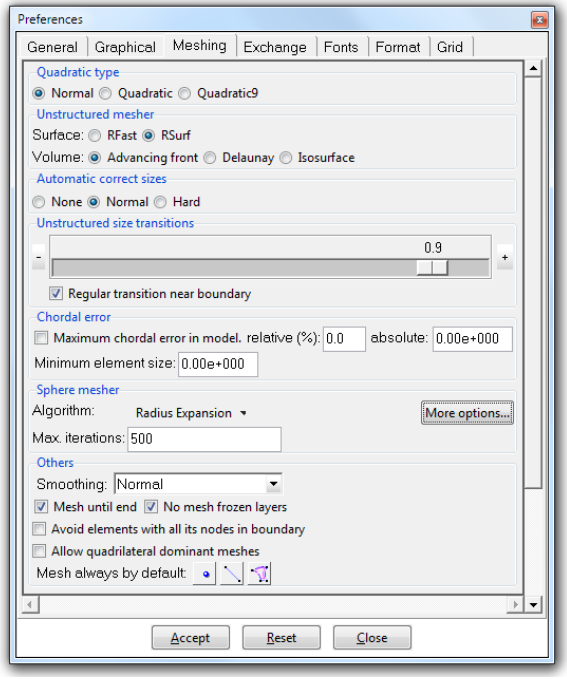


Figure F.5 Meshing preferences.

Click **Accept** and then **Close** the window.

It is recommendable to first generate a (triangular) surface mesh in order to generate a good (tetrahedral) 3D mesh, because this way we can check the mesh quality faster. The steps to generate a good tetrahedral mesh will be:

- Generating a surface mesh
- Check the quality of the mesh generated and make corrections, if necessary
- Return to geometry view and generate a geometrical volume out of the casting surfaces
- Generate a tetrahedral mesh of the casting, overwriting the previous mesh
- Check again for mesh quality
- Return to geometry and generate the mould volume, and then generate the final casting and mould mesh, overwriting the previous mesh; this is the only mesh needed; the previous ones were generated for verification

Save the project before meshing. Go to **File → Save as…**
Go to the Mesh menu as shown in ***Figure F.6*****11**:

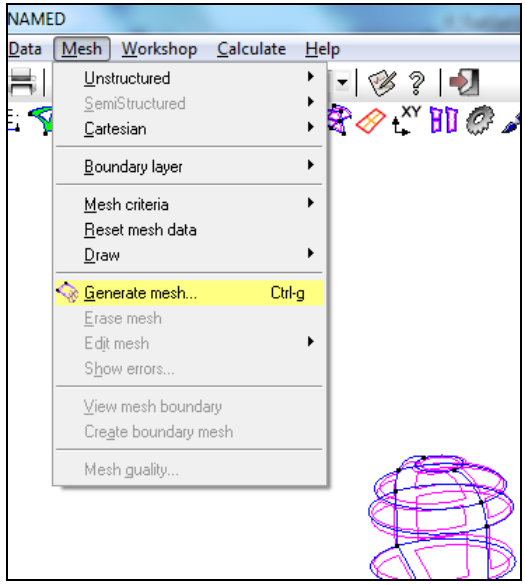


Figure F.6. Mesh generation screen.

The automatic assigned size of the mesh is 6; change this value to 2.5 and click **OK**. The result will be like this: ([4])

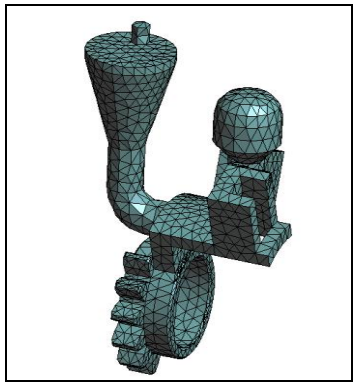
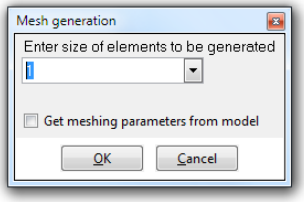


Figure F.7. Non-uniform triangle mesh.

We can see in ***Figure F.7*** that elements are not too uniform in order to be acceptable. Let's decrease the size of the elements to obtain a more uniform mesh.

Go to Mesh → Generate Mesh…
And set a mesh size of 1:



NOTE ([4])*: We can change between geometry view and mesh view by clicking on the icon:* *and also change normal view to render view by right-clicking on the geometry and selecting* ***Render →Flat*** *or* ***Render →Normal***

The result should be like in ***Figure F.8***:

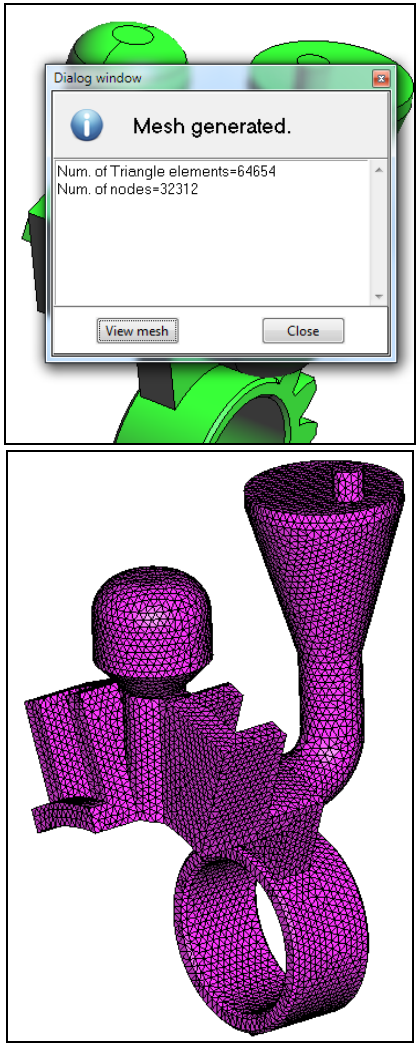


Figure F.8. Mesh generation.

Compare the results of ***Figure F.72*** and ***Figure F.8***, and look, for example, in ***Figure F.8***, the sizes of elements in the upper part of the casting, the small cylinder (entrance of material). We can see there how the mesh quality is improved.

Let's now return to geometry by clicking and generate the volume and tetrahedral elements. Go to Geometry → Create → Volume → By contour or click the **Create volume** icon .

The following will appear in the command line:
"Enter surfaces to define volume (ESC to leave)"

Select all the surfaces of the casting with a selection window ([5]):
"Added 108 new surfaces to the selection. Enter more surfaces. (ESC to leave)"
press **Esc**

"Created 1 new volume. Enter more volumes"
And now press **Esc** again:
"Leaving volume generation"

Now regenerate a mesh with size of 1 in order to obtain a mesh of the volume. The result will look again as in ***Figure F.8***.

Generating the mould mesh

Return to geometry view. Click on Layers icon, and turn on the layer **mould**. With the layer mould selected, double click on the layer mould.

Now, we will generate a volume for the mould.

Go to **Create volume**, and select all the surfaces with a selection window:

NOTE ([5]): We can use the Layers icon to facilitate the selection of the casting surfaces. Leave the **cast** layer on and the **mould** layer off for this selection.

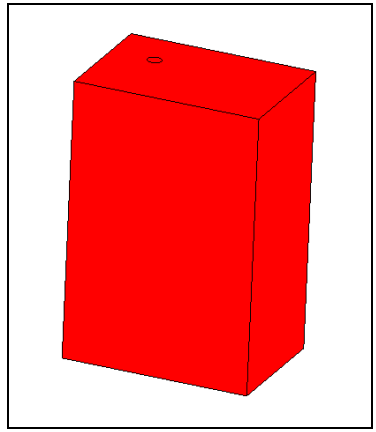


Figure F.9. Mould volume generation.

Now zoom on the small circle (material entrance) and click on the small surface in order to deselect it. The result should look like this:

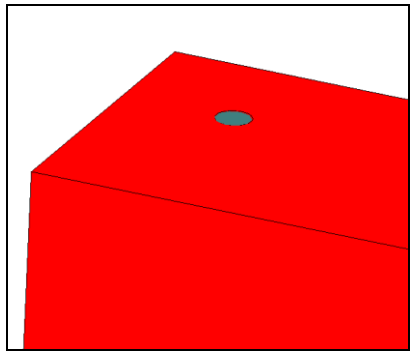


Figure F.105. Mould volume generation.

Now press **Esc** twice and the mould volume will be generated.

Assigning mesh size to the mould

In the case of the mould geometry, we will try to use tetrahedral elements as big as possible, in order to reduce the total number of finite elements of the problem. This will reduce the calculation time; so, instead of assigning an automatic size of 1 to all the geometry (cast + mould) we are going to do the following:

Turn off the layer **cast**.

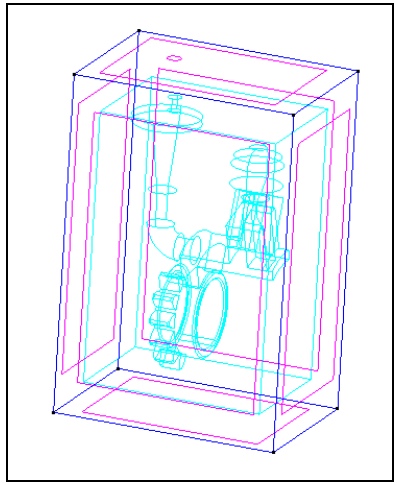
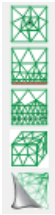


Figure F.11. Mould layer.

And now, we will see in detail how to assign properties to the mesh.

Using the **assign mesh sizes** icon, we will assign properties to the geometrical entities of our geometry:



We have to assign mesh sizes in geometrical hierarchy order: **Points, Lines, Surfaces and Volumes.**

Let's assign a mesh size of **7.0** to the points, lines and surfaces of the mould, and a size of **5.0** to the mould's volume.

a) Click on assign sizes to points, the first icon .

The following window will open:

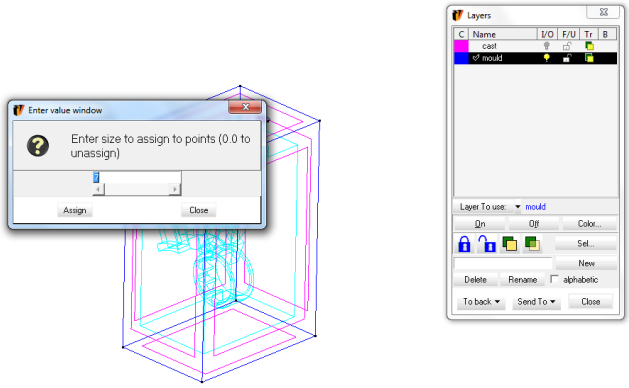


Figure F.12. Mesh generation

b) Assign a size of **7** in the window and click the **Assign** button.
c) Select all the points of the mould, with a selection window:

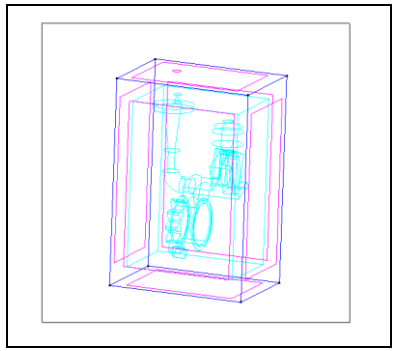


Figure F.13. Selection window

d) In the command line, the following message will be displayed:
"Added 8 new points to the selection. Enter more points. (ESC to leave)"

These are all the points added to the selection; the points are added, and as it says in the command line, we have to press **Esc** to accept the selection and leave.
The following message will appear in the command line:

"Assigned size=7 to 8 new entities"

e) click **Close** to close the enter value window.

Now we repeat **points a) to e)** for lines and surfaces, with the same **size of 7.**

Once done, we repeat **points a) to e)** for volumes with **size 5. We can use the button** to assign the sizes directly to points, lines and surfaces. (it will be assigned to all the active points, lines and surfaces)
Once the sizes are all assigned, we generate the mesh again (same general mesh size: 1). The result shall look like this (***Figure F.14***):

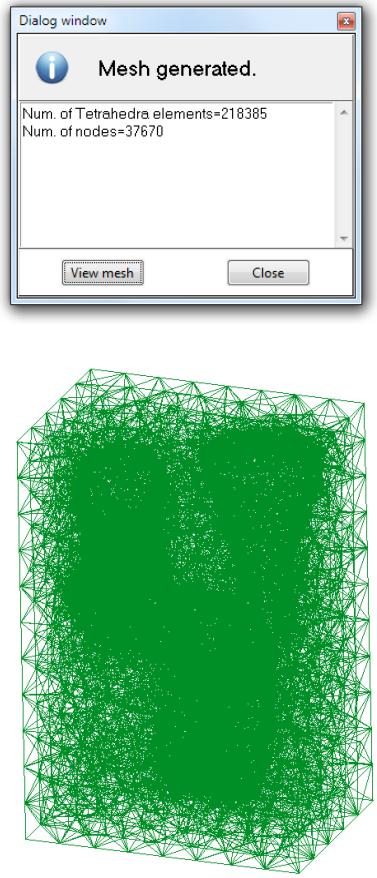


Figure F.14. Generated mesh

Now that the mesh is generated, we will see in detail how to set the process parameters for this particular type of problem, gravity casting.

We will continue the step-by-step process with the **Process bar**.

Step 4

FOUNDRY PROCESS DEFINTION

Until now we were dealing with geometrical parameters, definitions, etc.

From now on, we will deal with real foundry parameters, characteristics, processes, etc. It is very important to set the parameters properly. The closer the parameters to reality, the more accurate results. So read on and let's do casting simulation.

The first step we take once we have the casting and mould geometry defined and meshed, is let Vulcan know what type of process we are going to calculate. Click on **define the main process characteristic**, the 7th icon:

A window with 3 labels will appear. In the **Process type** label, select **Gravity,** as shown in ***Figure F.15***.

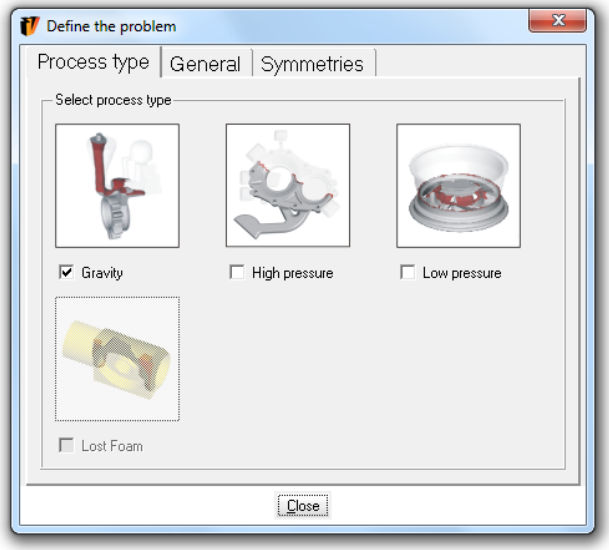


Figure F.150. The Process type window

Once this step is taken, Vulcan adjusts all the internal parameters and the remaining icons to this particular type of process.

In the **General** label, we have a set options to define: units, gravity direction (with respect to the coordinate system compass), environment temperature, etc.

Set the gravity direction to –Z, and the Environment temperature to 25 Degrees Celsius, as shown in ***Figure F.16***:

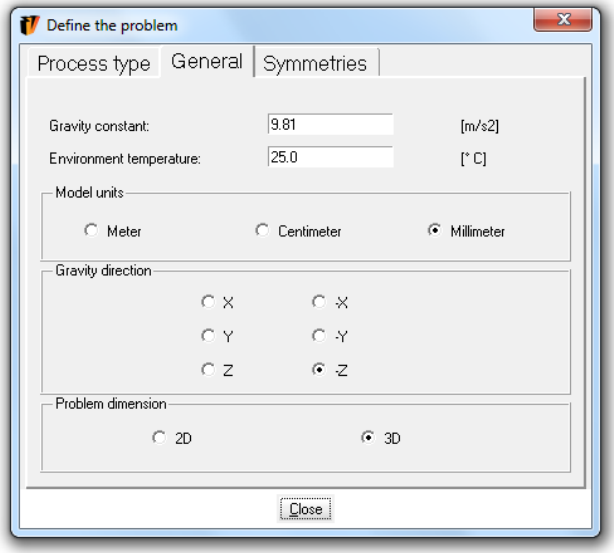


Figure F.16. The Process type – General window

In the **Symmetries** label (***Figure F.17***) we have to define the symmetry planes created; if there are no planes of symmetry, simply leave No planes selected:

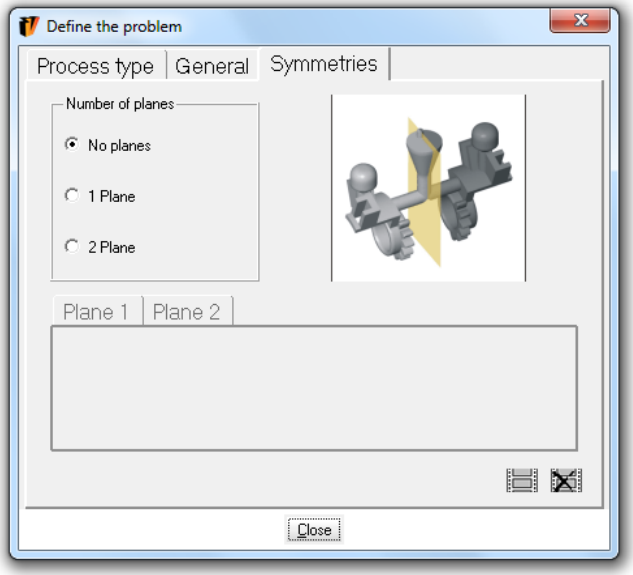


Figure F.172. The Process type – Symmetries window

Now the gravity casting process is defined; click **Close** ([6]) to continue to the next step.

NOTE ([6]): *All the changes made will be saved upon clicking on the **Close** button.*

Step 5

FOUNDRY COMPONENTS DEFINTION

Click on the **Components definition** icon:

The following window will open (***Figure F.18***):

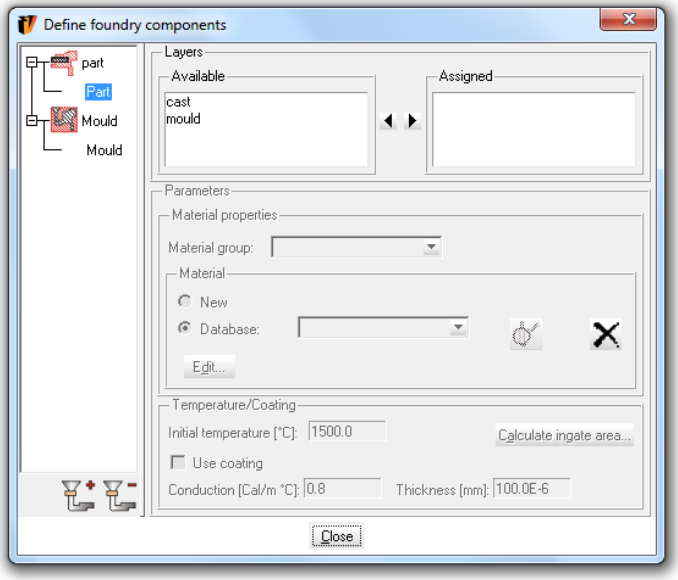


Figure F.18. Foundry components definition.

First we have to assign each layer to the corresponding foundry component. On the example shown, the Part component is highlighted on the tree. In the **Available** window, select **cast** and click on the Icon. The **cast** layer will move to the **Assigned** window.

Now, select the Material group and specific material from the database, and set a temperature of 1500 Degrees Celsius, as shown in ***Figure F.19***:

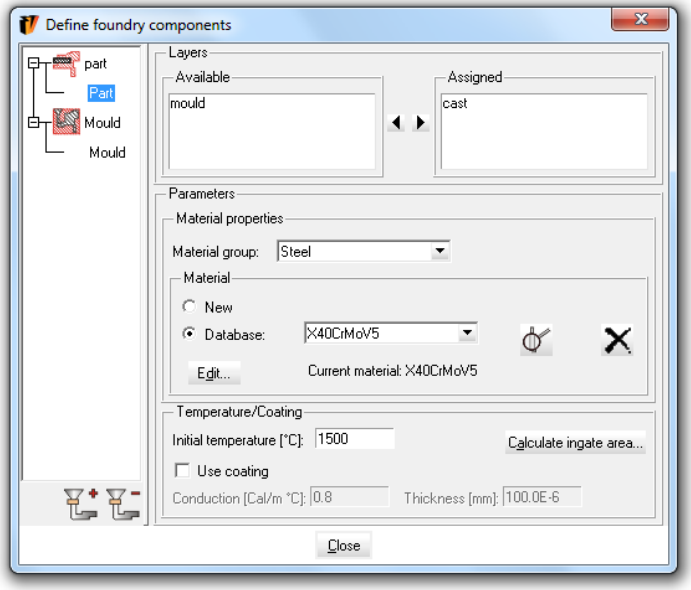


Figure F.19. Foundry components definition.

Now perform the same procedure for the mould, first selecting the **Mould** component on the tree, and then **mould** from **Available** window. Again, it should move to **Assigned** window.
Assign in the **Mould materials** group (permanent or sand mould), the **sand** material at initial temperature of 20 Degrees Celsius:

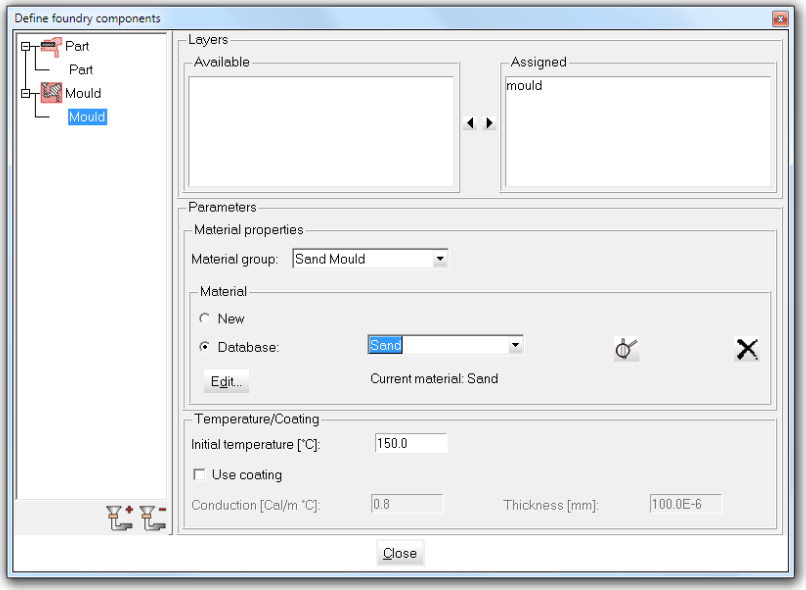


Figure F.20. Foundry components definition.

Click **Close** to save the changes.

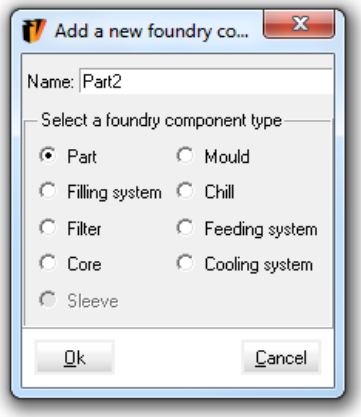
***NOTE**: Let's explain some of the icons that appear in the **Components definition** window. You can skip this note to continue with the example, and go to the* Step 6*.*

Click again on the icon:

In case of having multiple layers, you can assign and unassign layers to foundry components by using the *icons (for example, in case of having the mould divided in upper part and lower part on different layers, or in case of having to make changes).*

Now click on the ***Add new foundry component*** *icon:*

The following window will appear:



This window has to be used in case of need to insert more components to the tree, such as more parts of the mould, or cores. On the example shown, there are only two layers, the whole mould and the part, so there is no need to create a new foundry component.

*In **Figure F.20** we can also see the **Use coating** checkbox. If we use this option, we can change the conductivity and the thickness for the specific coating we are using.*

*Now click on the **Edit…** button; this window will open:*

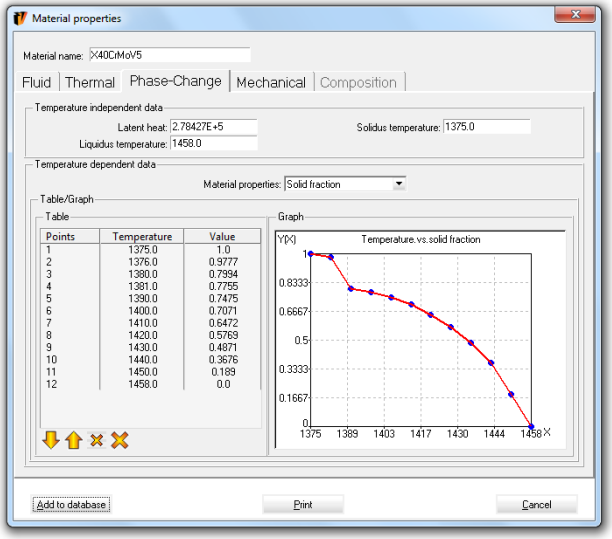


Figure F.21. Editing material properties

*Here we can see the material properties in both table and graphical views. We can change these data, add new materials, etc. See the Tutorials for more details. Now click **Cancel** on this window.*

*And click **Cancel** to continue with the example.*

Step 6

OPERATIONS DEFINITION

Now we are going to define the specific operations to be simulated. Let's click on the **Operations** icon:

The following window will appear (***Figure F.227***):

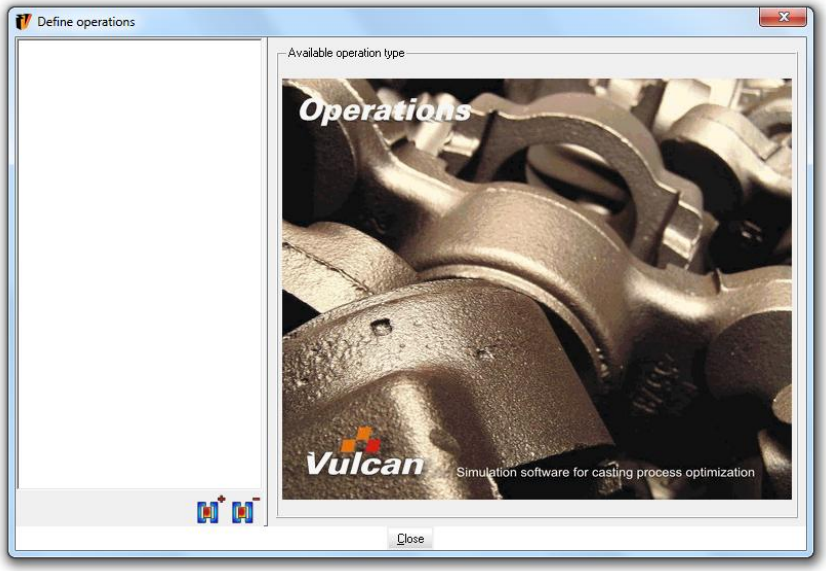


Figure F.22. Operations definition.

From the point of view of casting simulation, we can divide the casting process into three parts: **filling**, **solidification**, and **cooling**.

The filling begins when the material starts to enter into the mould, and finishes when the molten metal fills the entire mould.

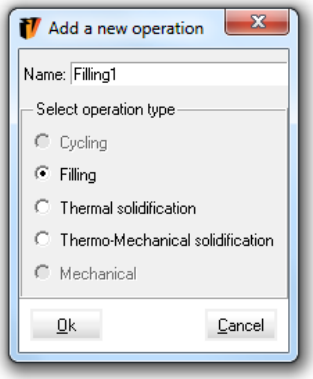
At this moment, solidification starts, as metal begins to cool down and solidify; solidification finishes when the last part of the casting turns into solid phase.

At this moment, cooling starts, and goes all the way down until the casting and mould reaches room temperature.

In our gravity casting example, we are going to run a simulation the **filling** and **solidification** parts of the process.

Click on the add operations icon:

The following window will appear:



Here we have to select all the operations we want to simulate one by one:
Leave Filling and click OK, and then enter on the same previous icon, and select **Thermal solidification** operation.

The result should look like this:

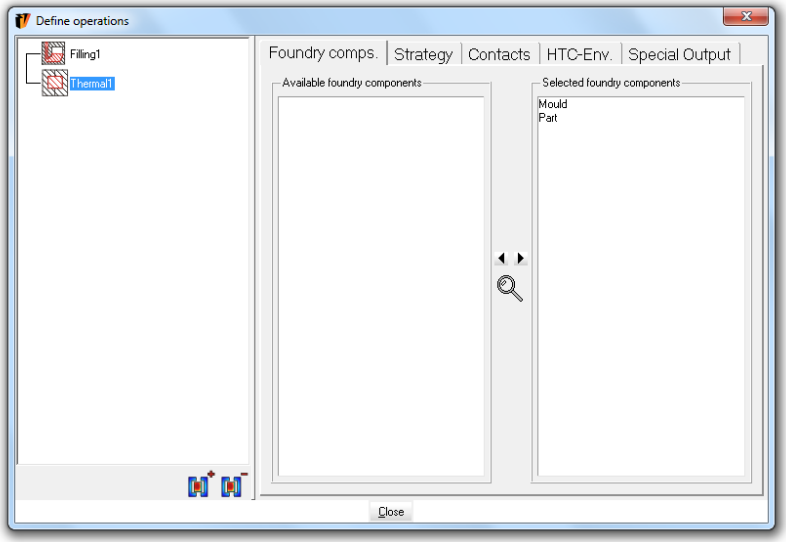


Figure F.238. Operations definition.

In ***Figure F.23*** we can see that all the foundry components are assigned by default to the filling and solidification operations. In (***Figure F.18***) we created those foundry components. Here we have the option of changing the foundry components, selecting different components for different operations, etc.

To continue with the example, we leave the default selection, and then, we are going to view and set all the parameters for both operations. We have to go trough all the labels and change the default parameters for those of interest. In our example, we will see all the options and a short explanation of the principal parameters.

In strategy label (***Figure F.24***), we set the Flow rate for gravity filling (default: 2 dm^3/Sec), and we decide whether to use tilt pouring or not. In case of using tilt pouring, we have to select it and enter all the options. Change flow rate to 0.4 dm^3/Sec, and continue (Click) to the next label.

We also can calculate by filling time or by gravity (in this case we have to assign the height between the spoon and the mould.

Another parameter to assign is the Mould Surface Finish. We have 4 values to decide the roughness of the mould, between low and high.

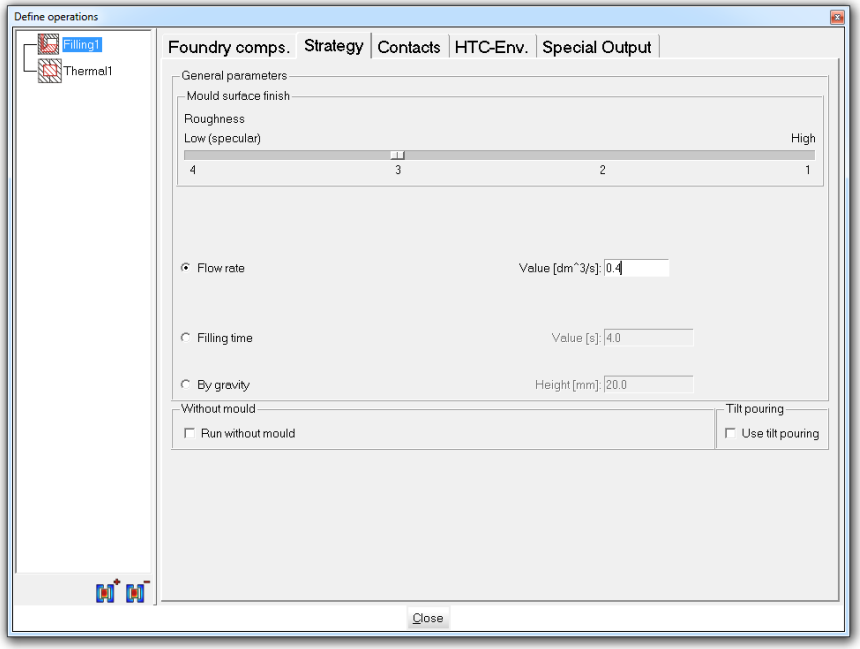


Figure F.24. Strategy label

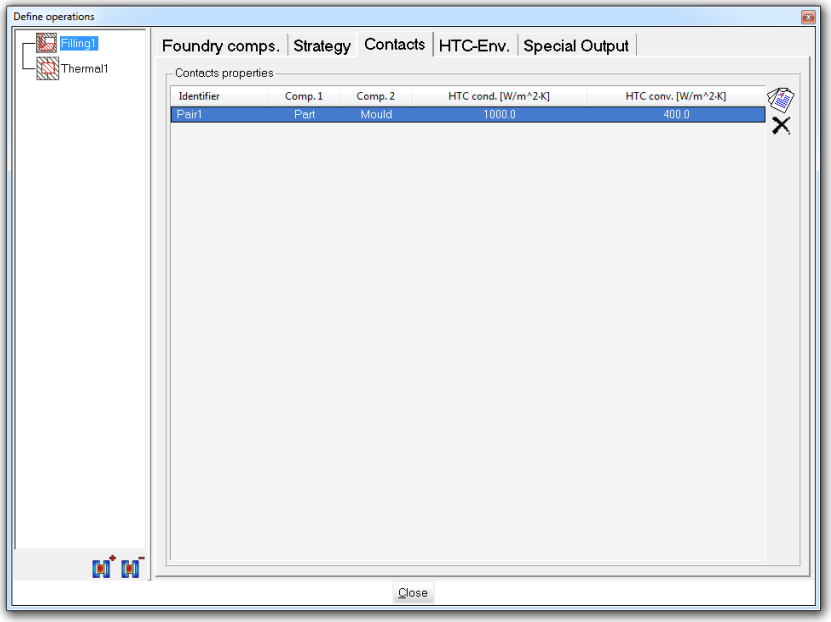


Figure F.25. Contacts label

In the Contacts label (***Figure F.25***), we set the values of the Heat Transfer Coefficients (HTC) for the contact between the casting and the mould. This contact is characterized by two different values: HTC conduction and HTC convection. Change values to:

HTC conduction: 800
HTC convection: 250

And continue to the next label.

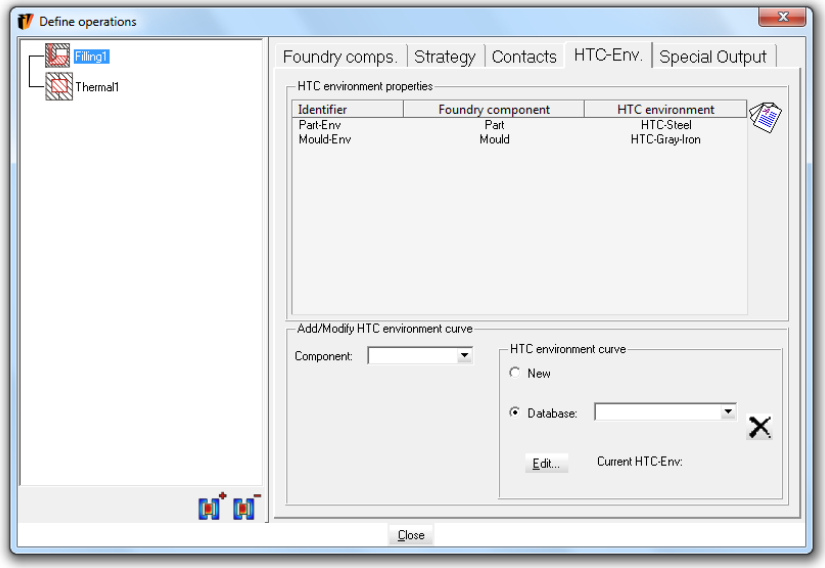


Figure F.26.HTC-Environment label

Let's have a look on ***Figure F.26***. Here we set the value of heat transfer coefficient with environment (HTC-Env) in this case, environment is the air at the temperature set on the problem definition ([7]).

To continue with the example, leave all the default options and continue to the next label.

NOTE ([7]): We can modify or create a new value for this parameter by entering on the Edit... button and setting a new value. The parameter could change, for example, if the air is not still (there is air circulation with fans)

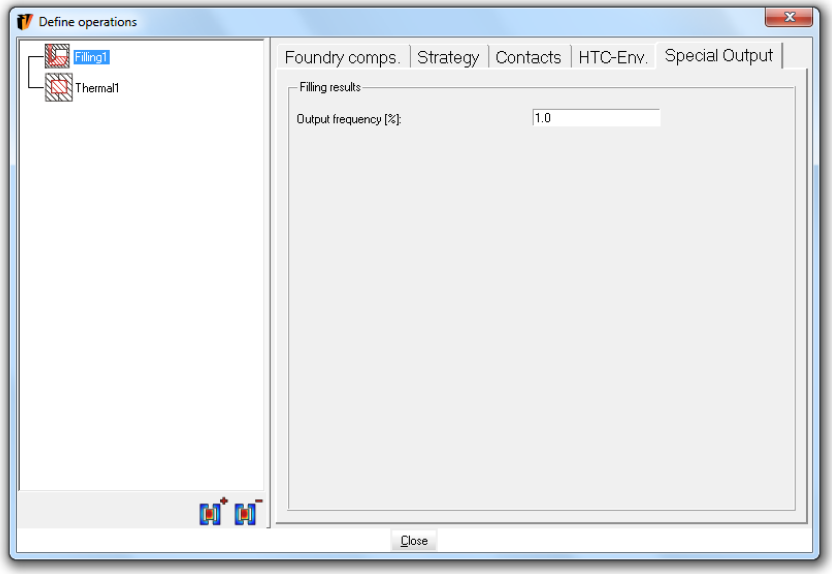


Figure F.27. Special Output label

In ***Figure F.27***, we define the number of results to be written during the analysis. Leave the default option.

Now, change to **Thermal1** operation on the operations tree, as shown in ***Figure F.28***:

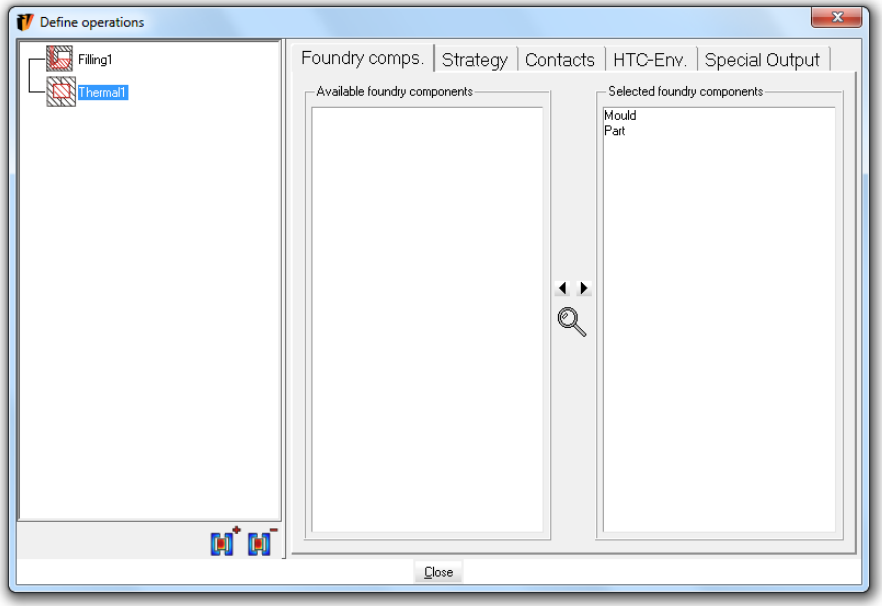


Figure F.28 ***Thermal1*** *settings*

In this screen, foundry components are already selected. Go to the next screen:

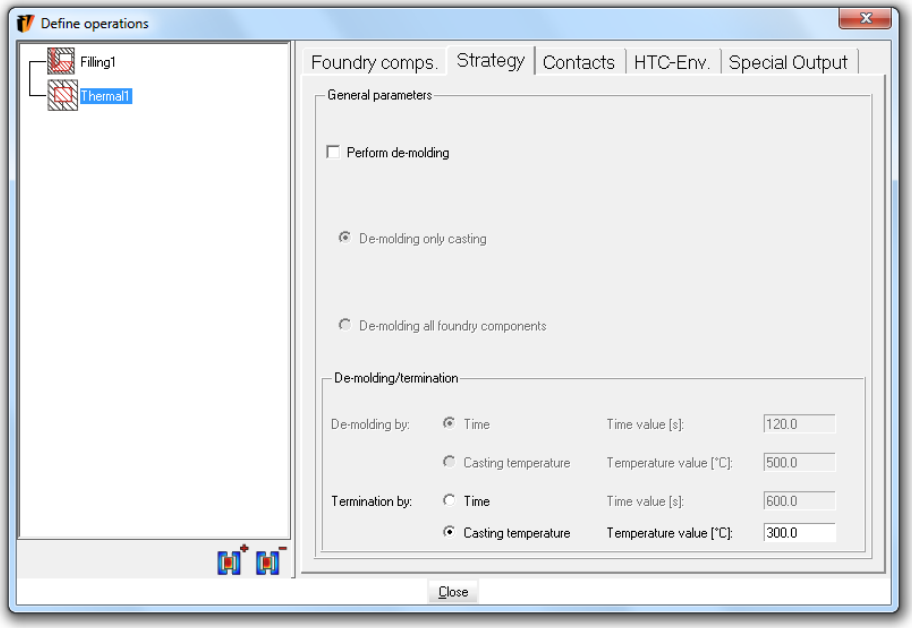


Figure F.29. Thermal Strategy label

In this screen, we define the **De-moulding** and **Termination** characteristics of our process. In the current problem, we leave de-moulding deselected, and we select termination by Casting temperature, 300 ºC. Click on the next label.

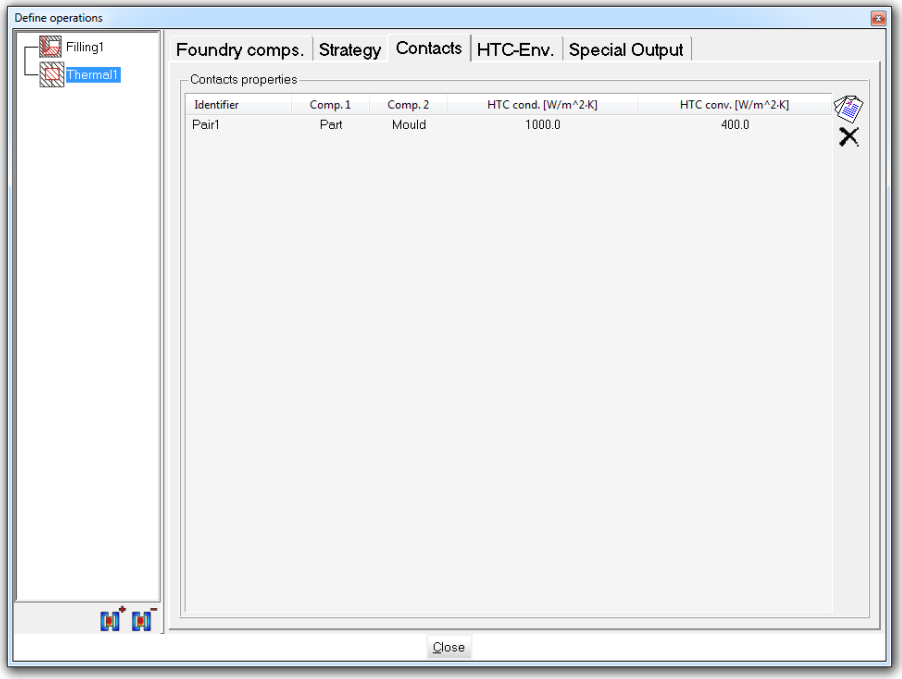


Figure F.30.Thermal Contacts label

Here again we change the HTC conduction to 800, and HTC convection to 250, and continue to the next label.

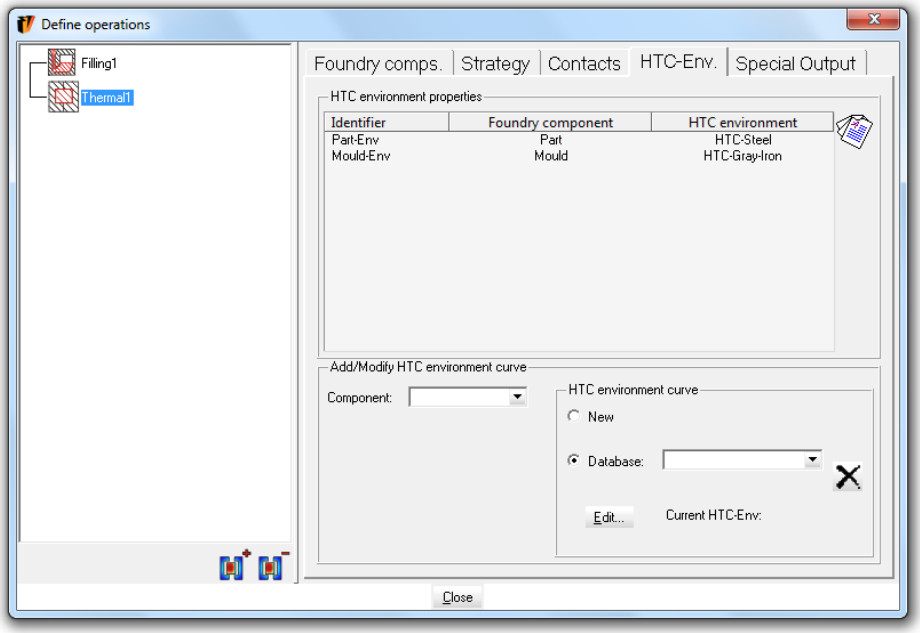


Figure F.31.HTC-Environment label

We have pre-selected the same options as for the filling analysis. Continue to the next label.

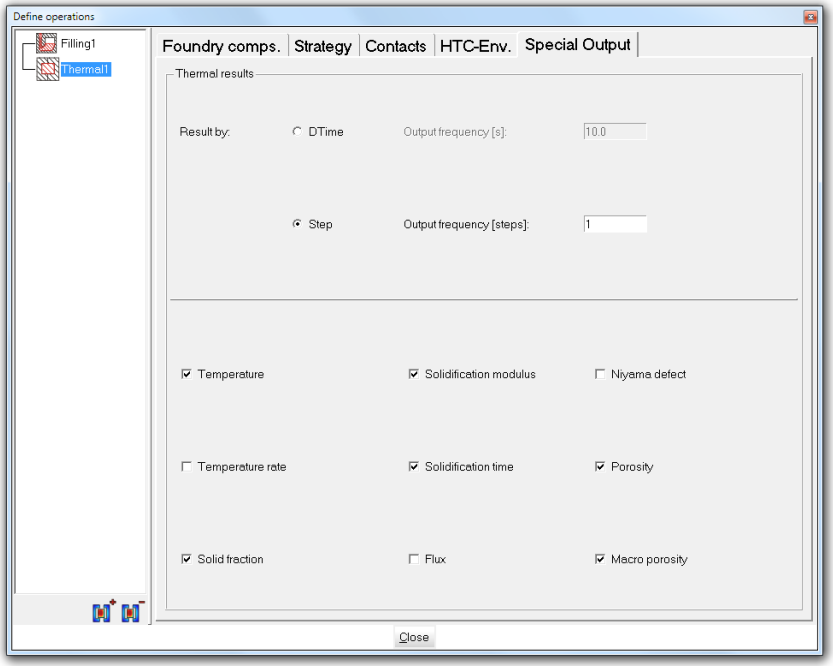


Figure F.32.Thermal Special Output label

In this label, we define the thermal results that we are going to visualize on the post-process. In the example, leave Temperature, Solid fraction, Solidification modulus, Solidification time, Porosity and Macro Porosity.

Now that we have defined all the problem parameters, click on the **Close** button. The parameters will be saved.

Step 7

SAVE PROJECT

Before start the calculation, don't forget to save the project:

Click on **Save** icon.

NOTE: the filename must not contain spaces or symbols.

Step 8

CALCULATION

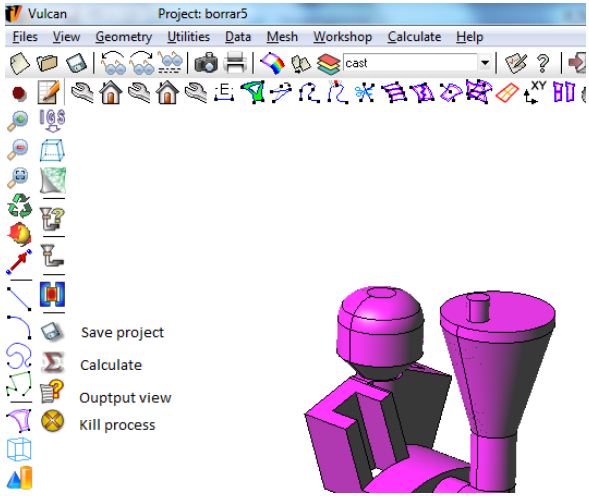


Figure F.33.Calculate, Output view and Kill Process icons.

Now, following the column of icons shown in ***Figure F.33***, we first saved the project and then **Calculate** it. The calculation time will depend upon the complexity of the problem.
The calculation window will look like this:

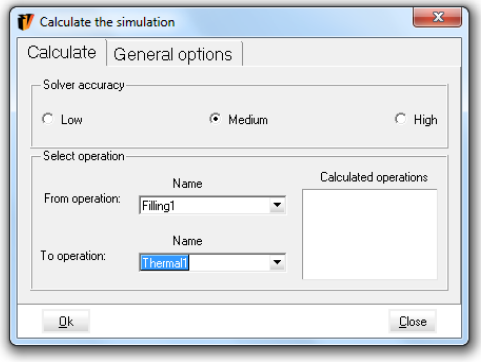


Figure F.39.Calculation window

Select **From operation** filling **To operation** Thermal1 and click OK. The calculation will start:

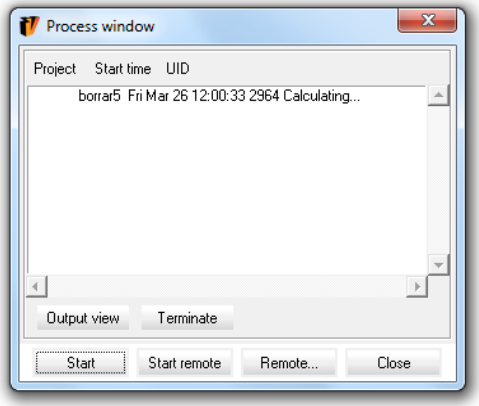


Figure F.34.Calculation window

We can view the **Output** view by clicking the icon and see the calculation status, and also interrupt and cancel the calculation by **Terminate** the process, make changes and **Calculate** again. Close the window.

OUTPUT VIEW

Once the window is closed, we can still access to the output view by clicking on the icon:

And we will see the calculation process:

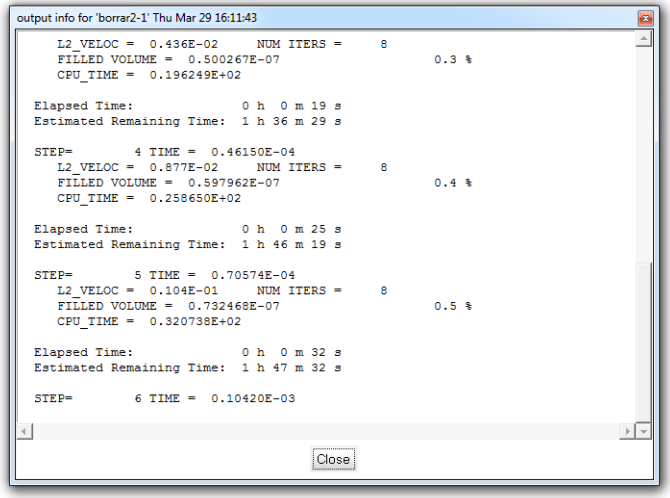

output info for 'borrar2-1' Thu Mar 29 16:11:43
L2_VELOC = 0.436E-02 NUM ITERS = 8
FILLED VOLUME = 0.500267E-07 0.3 %
CPU_TIME = 0.196249E+02
Elapsed Time: 0 h 0 m 19 s
Estimated Remaining Time: 1 h 36 m 29 s
STEP= 4 TIME = 0.46150E-04
L2_VELOC = 0.877E-02 NUM ITERS = 8
FILLED VOLUME = 0.597962E-07 0.4 %
CPU_TIME = 0.258650E+02
Elapsed Time: 0 h 0 m 25 s
Estimated Remaining Time: 1 h 46 m 19 s
STEP= 5 TIME = 0.70574E-04
L2_VELOC = 0.104E-01 NUM ITERS = 8
FILLED VOLUME = 0.732468E-07 0.5 %
CPU_TIME = 0.320738E+02
Elapsed Time: 0 h 0 m 32 s
Estimated Remaining Time: 1 h 47 m 32 s
STEP= 6 TIME = 0.10420E-03
Close

Figure F.35.Output view.

TERMINATE PROCESS

If we want to change parameters and calculate again, we would click on the last icon:

This icon will terminate the process. In the example, we will leave the computer calculating and wait for the end of calculation. We will not click on the icon.

Exiting Vulcan while is still calculating

When the pre-process is over and we are calculating a process, we can exit Vulcan application but without interrupting the calculation:

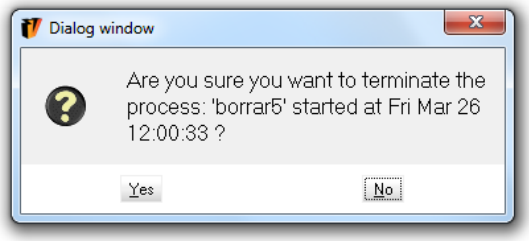


Figure F.36. Exiting Vulcan while is still calculating.

Click **No** and Vulcan will exit. We can open Vulcan later, during or after the calculation. The computer, of course, must remain on.

Step 9

POST-PROCESS

Post-process means the visualization of the simulation results; this is perhaps the most important step of all the casting simulation steps; here is where we read and analyze the simulation results.

Once the calculation is finished, we can visualize the results of this simulation. There are many different ways to do it, like colour maps, vectors, iso-surface curves, virtual thermocouples, etc.

Figure F.37 shows the window that appears once calculation step is finished; we can directly access to post-process by clicking on **Postprocess** button.

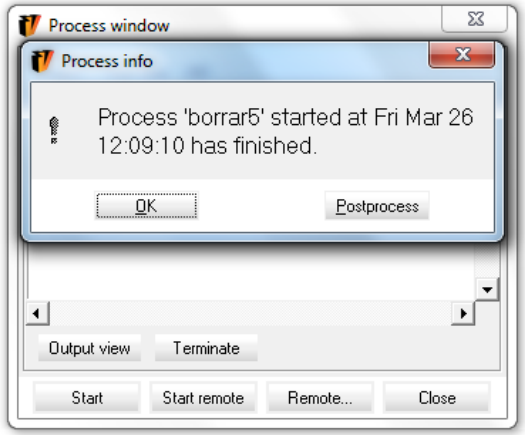


Figure F.37.Process info.

Another way to switch between pre- and post-process is by clicking the **toggle** ([8]) icon .

NOTE ([8]): Every time we toggle, when we return to post-process, we have to load the results.

The first step in results visualization is to load results of any **foundry operation** (Filling, thermal solidification, thermo-mechanical solidification). Let's load the filling operation and analyze the results.

Go to **Vulcan results** → **Load results**… or select

And select the **Filling1** operation:

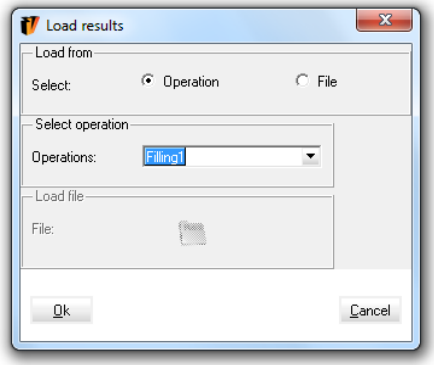


Figure F.38. Results loading

Click **Ok** and the filling results will be loaded.

Now that we have all the filling results loaded, let's begin, for example, to see the evolution of the casting material in the mould; where there could be air entrapment; or the advancing fronts of material could encounter; the last spots of the mould to be filled, etc.

To see this, we will use a tool called filling mater. In this animation, we will see the material free surface in red, the filled areas in grey, and the mould will be transparent.

Go to main menu:

Vulcan results → **Filling mater** or press the button
Once the Animate window appears, press the **Play** button:

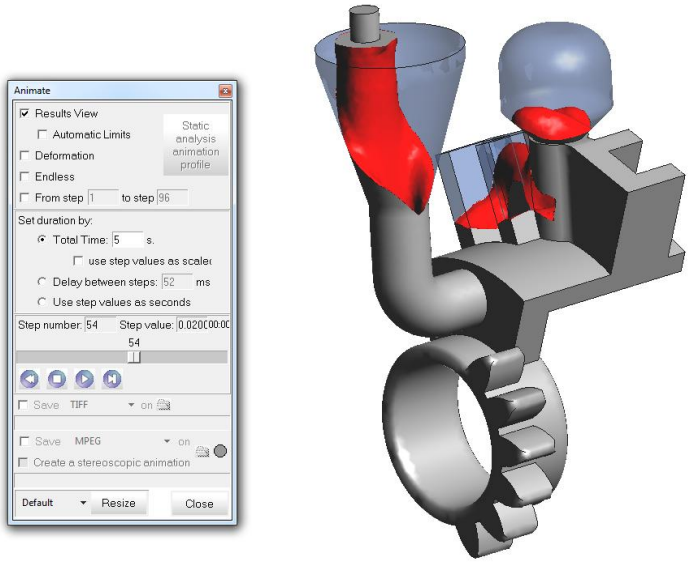


Figure F.39 Filling mater animation.

In ***Figure F.39*** we can see the **Animate** window and the control buttons. We can keep this window open while we rotate, zoom in and zoom out the casting. Let's explain the basic Animate controls.

Basic Animate window controls

Automatic Limits: GiD searches for the minimum and maximum values of the results along all the steps of the analysis and uses them to draw the results view through all the steps. Previously, it needs to visualize some resutls.

Deformation: Permits to record animation with a specific deformation.

Endless: The animation continues indefinately.

Total time: specfies the duration of the clip.

Delay: Specify a delay time between steps in milliseconds.

Use step values as seconds: the number of step will be used to the duration of the clip.

From step I to step K: allows the user to do an animation between step number I and setp number J , both included. This is also useful to skip the first step of an animation of a deformed mesh with a result visualization, which is the original state of the mesh, without deformation and without the result visualization.

Save **TIFF/JPEG/GIFs** on: Save snapshots, in TIFF, JPEG or GIF format, of each step when the 'Play' button is pushed. Here the filename given will be used as a prefix to create the TIFF/JPEG/GIFs; for instance, if you write MyAnimation, TIFF/JPEG/GIF files will be created with names MyAnimation-01.tif/MyAnimation-01.gif, MyAnimation-02.tif/MyAnimation-02.gif, and so on.

Save **MPEG/Avi mjpeg/AVI** MS Video 1/AVI raw True Color/AVI raw 15 bpp (VD) /AVI raw 16 bpp (MS)/GIF on: Entering a filename here, a MPEG/AVI/GIF file will be created when 'Play' button is pushed.

Then we have the animation controls:

these controls are the standard Play/Pause/Stop/Fast Forward/Rewind controls. We can control the evolution of the animation with them.

Let's say now that we want to see other phenomena, for example, turbulences formation, highest filling velocities inside the casting, velocities at which the advancing fronts encounter, etc. We will use a different representation to see these results.

Go to:

Vulcan results → **Filling vectors** or press the button

And again, the animate window will show up. The animation will show the velocity vectors inside the casting:

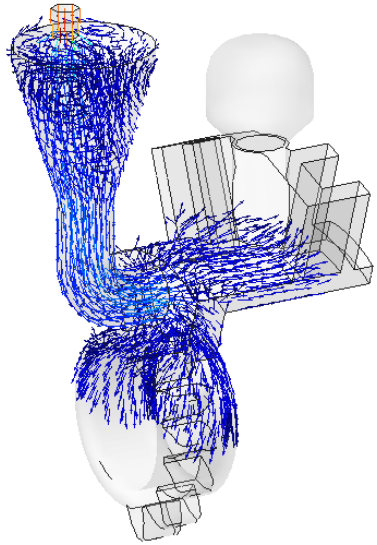


Figure F.40. Filling vectors animation

The colour scale represents the module of velocity. In ***Figure F.40*** we can see the highest velocities in red, and the lowest velocities in blue.

Once finished with filling vectors, go to:

View results → **No results**

Now we want to see parameters like temperature of front encounters, early cooling during filling, and possible solidification during filling.

To see this kind of results, we will go to:

Vulcan results → **Temperature evolution** or press the button

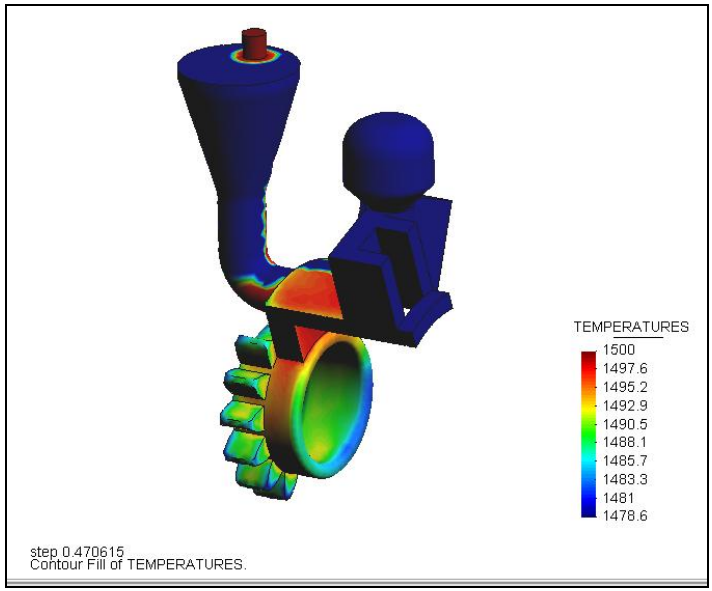


Figure F.41. Temperature evolution during filling.

Here we can see the temperatures and the temperatures colour scale, and also the time step since the start of filling.

Now we will load the second **foundry operation** we had run: solidification.

Solidification goes from the end of filling until the casting is completely solid.

Let's see the results. Go to:

Vulcan results → **Load results…** and load the **Thermal1** operation:

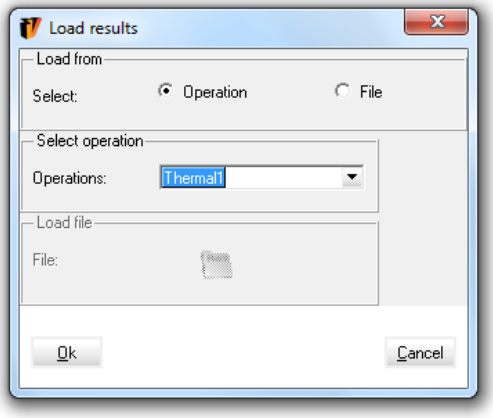


Figure F.42. Thermal operation loading.

Once the results are loaded, click Ok and let's see some solidification results.

This time we will see temperature evolution in both casting and mould, excess temperature on the cores, etc.

Go to:

Vulcan results → **Temperature evolution** or press the button

The result will be like this:

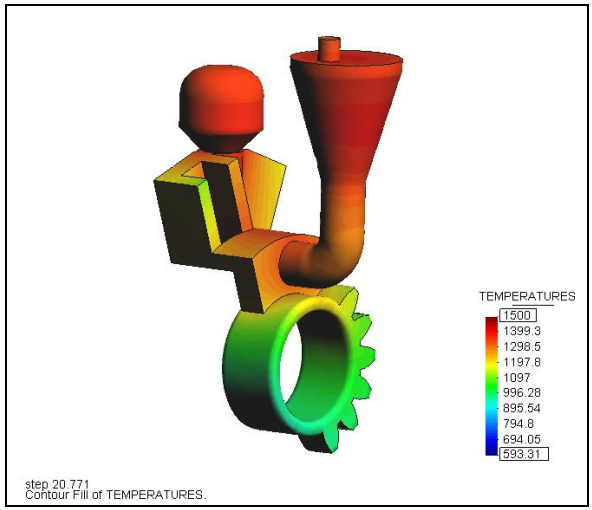


Figure F.43. Temperature evolution.

In this result, we can animate and see the temperature evolution in the casting surface, but it would be interesting to "cut" the casting and see the temperature inside.

To do so, we will make a cut of the casting and mould.

Go to:

Vulcan results → Temperature in cut…

Select **Cut xz plane** button, and then draw a line dividing the mould horizontally in two equal halves. The result will be:

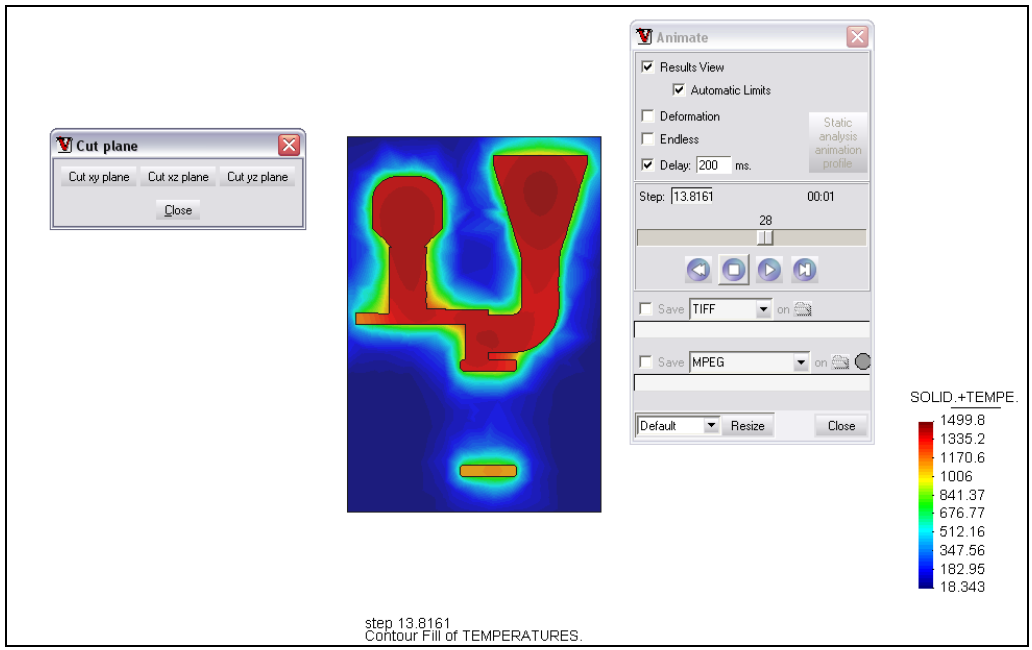


Figure F.44. Temperature in cut.

On the figure, we can see the temperature evolution of both the casting and the mould, in the **Cut xz plane**. By pressing the **Play** button on the animate window, we can see all the temperature evolution during solidification.

Now, we will add a virtual thermocouple in a point in the middle of the casting.

Go to:

View results → Graphs → Point evolution → Temperatures

And select a point in the middle of the casting:

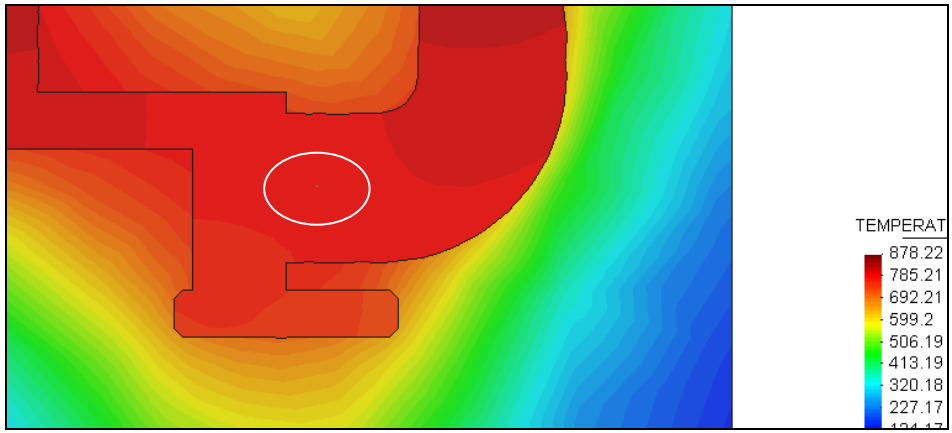


Figure F.45. Point selection.

To select the point, first we click on **Select Nodes** in the **right menu** on the Vulcan screen:

because we will plot the temperatures in a node of the mesh; then, we have to click once in the zone shown in ***Figure F.45***, for example. Once there is a red point signalled, we press **Esc**.

The result will be:

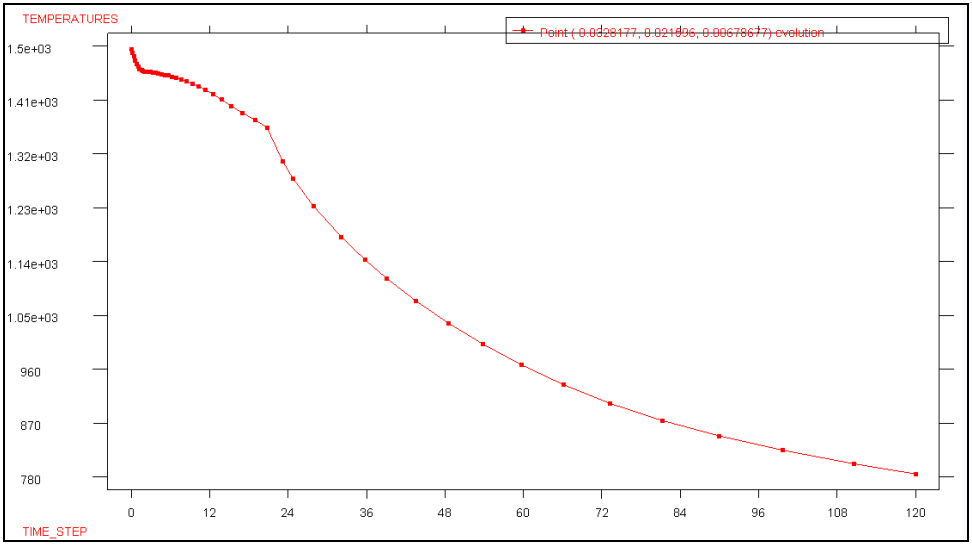


Figure F.46. Virtual thermocouple in the casting.

Once finished, to **View results → No graphs** and then, **View results → No results**

In the previous temperature images we cannot see the casting phase change, from liquid to solid. In order to see this and, for example, the effectivity of our riser system, the evolution of the solid phase, the last points in the casting to reach solid state, etc, we will do the following:

Go to:

Vulcan results → Solid fraction evolution

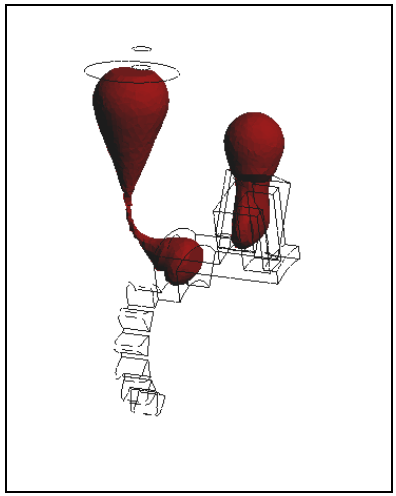


Figure F.47. Solid fraction evolution.

Here we have the liquid fraction of the casting is plotted in red, and the solid part of the casting disappears while is forming, so we only see the remaining liquid parts inside the casting as it solidifies, during the animation. The last points remaining in red should be the risers.

Finally, we will plot the **solidification modulus** of our casting. The modulus is the volume-to-surface ratio of the casting, and it can also be seen inside the casting by making cuts. Let's go back to ***Figure F.43*** and make again the temperature evolution, and then again temperature in cut. Once we have the results like in ***Figure F.44***, go to:

View results → Contour fill → Solidification Modulus

The result will be:

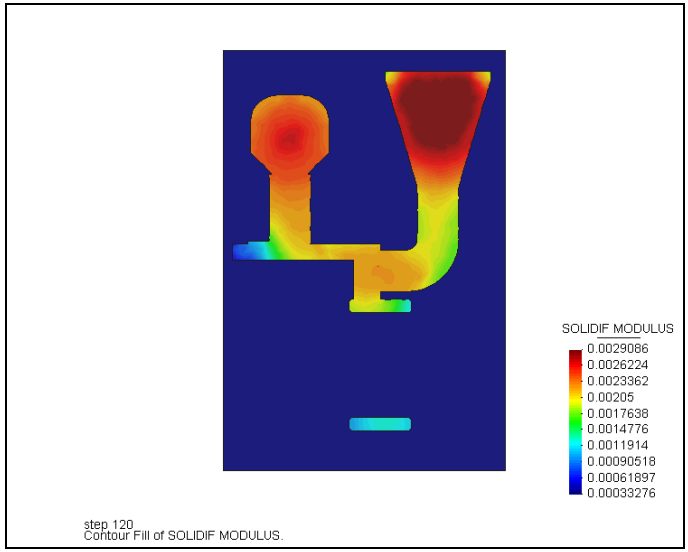


Figure F.48. Solidification modulus.

G. HPDC

HIGH PRESSURE DIE CASTING SIMULATION USING VULCAN

Version 10.0

[This page is intentionally left blank]

INTRODUCTION

In this tutorial, we will follow an entire step-by-step procedure in order to run a **high pressure die casting simulation** ([1]) using Finite Element Analysis with Vulcan.

We will import an **IGES** geometry into Vulcan's graphical interface, mesh it, set the process parameters, and finally get simulation results (temperatures, velocities, turbulences, porosity defects, etc.)

Basically, a **Finite Element Analysis** consists of three phases:

- Pre-process
- Calculation
- Post-Process

During the **pre-process** phase, we will completely define the problem geometry, set the types of analyses to perform, and establish the physical properties of the materials, temperatures, etc.

Once the problem is completely defined, we will launch the **calculation**; the total calculation time will depend on the complexity of the problem (this is, the number of **elements** in both the casting and the mould). During the calculation time, user's participation is not required. The calculation will let us know when is finished.

When calculation is finished, the user can start the **post-processing** phase; this is the results loading, view and analysis; this is the objective of the finite element simulation.

This tutorial consists of 9 steps, that will guide you trough all the HPDC casting simulation process.

Before we start, let's describe Vulcan interface. You can skip Step 0 if you have previously worked with this interface.

NOTE ***([1])****: The term "**high pressure die casting**" includes both **hot chamber process** and **cold chamber process** in the diecasting nomenclature, but not the process called **low pressure die casting**, which is treated separately in the next tutorial.*

Step 0

THE PROCESS BAR

Vulcan has a process bar ([2]), that will guide us trough all the preparations, settings and calculation of our casting process. This process bar is normally located on the left side of the screen:

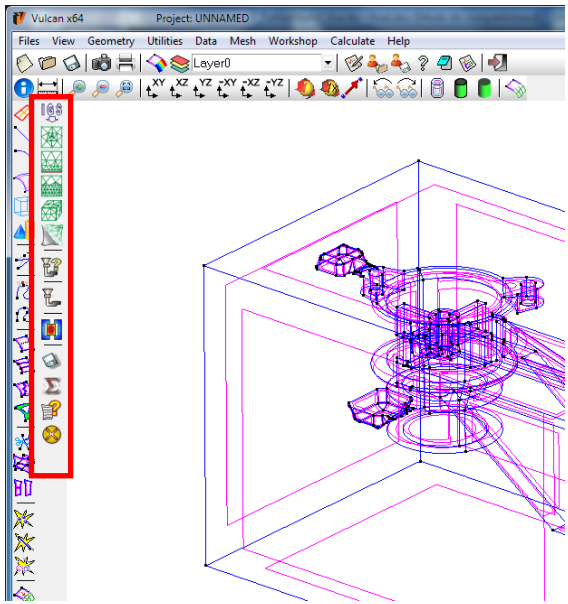


Figure G.1. The Process bar highlighted in red

NOTE ([2])*: If not present, you can activate the Process bar from the menu Utilities → Tools →Toolbars and also change its position to top, bottom, right, etc.*

Process bar icons

1. IGES read
2. Collapse entities
3. Mesh properties
4. Define the main process characteristic
5. Define foundry components
6. Define operations
7. Save the project
8. Calculate
   - 8.1. Process info
   - 8.2. Kill the process

These icons are placed in a logical order:

1 to 6	Pre-process
7	Save the project before calculate
8	Calculation

We can completely define and calculate the casting component under study using this bar, and in case necessary, interrupt the calculation by clicking icon 8.2.

Let’s begin by opening Vulcan and then with the step-by-step guided tutorial.

On the computer Desktop, double-click on the **Vulcan** icon:

the main screen will show up:

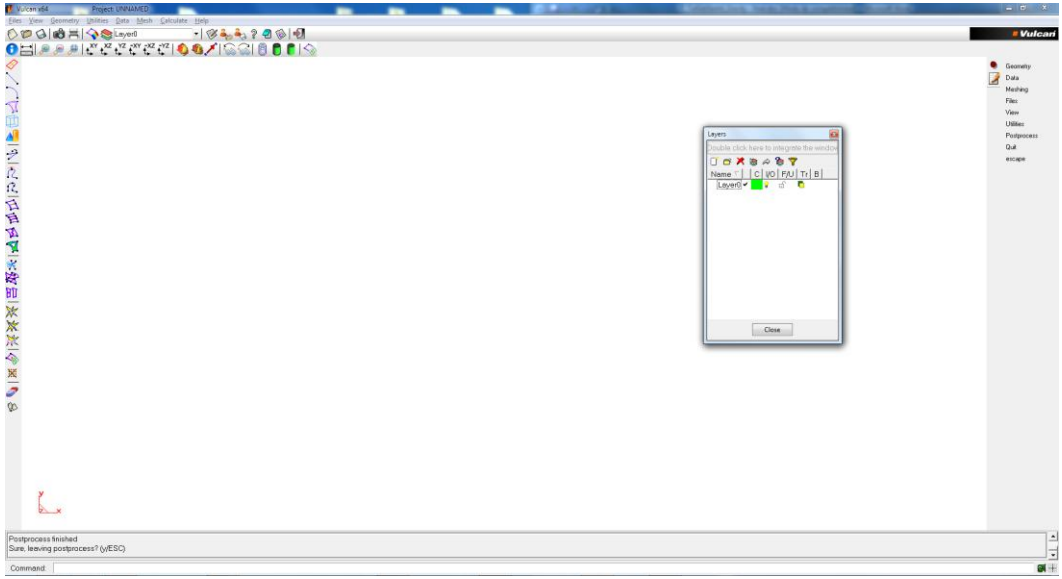


Figure G.2. Vulcan main screen.

Once Vulcan is opened, go to the main menu and click on:
Data → Problemtype → Vulcan

The following window will appear (***Figure G.3***):
(If the window doesn't appear, you can go directly to Step 1)

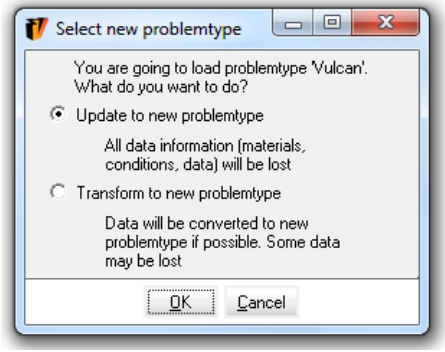

Select new problemtype
You are going to load problemtype 'Vulcan'.
What do you want to do?
Update to new problemtype
All data information (materials, conditions, data) will be lost
Transform to new problemtype
Data will be converted to new problemtype if possible. Some data may be lost
OK
Cancel

Figure G.3. Vulcan problemtype window.

Click OK and the process bar will open automatically; it will be used and described in detail in the present tutorial.

Now that we have the Vulcan pre-process interface opened and set, let's begin with the first step.

Step 1

READ AN IGES FILE

Before reading the IGES file, we are going to set Vulcan parameters in the optimal configuration for importing this type of files. In the main menu, click on Utilities → Preferences…
The following window will open:

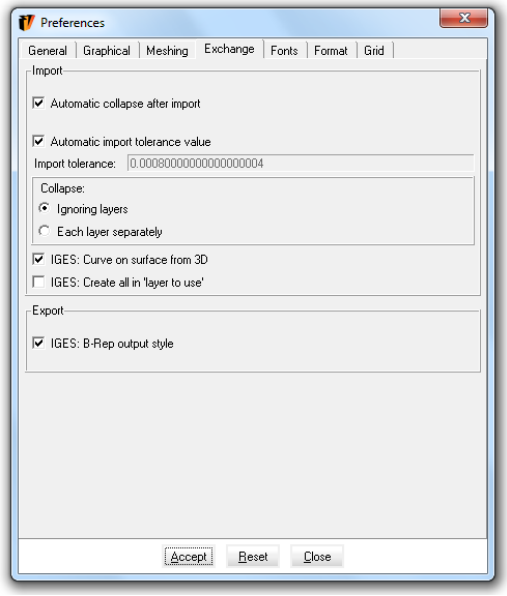


Figure G.4. Setting importing preferences.

The following checkboxes must be selected in ***Figure G.4***:

- ☑ Automatic collapse after import
- ☑ Automatic import tolerance value

Accept the changes and **Close** the window.

Now we are ready to import IGES files. Let's import a sample file.

Click the first icon on the process bar:

and read the following file:

"part_hpdc.igs" and click **Open**.

This file can be downloaded from the VULCAN web page http://www.quantech.es/support-vulcan.aspx

the result should look like this:

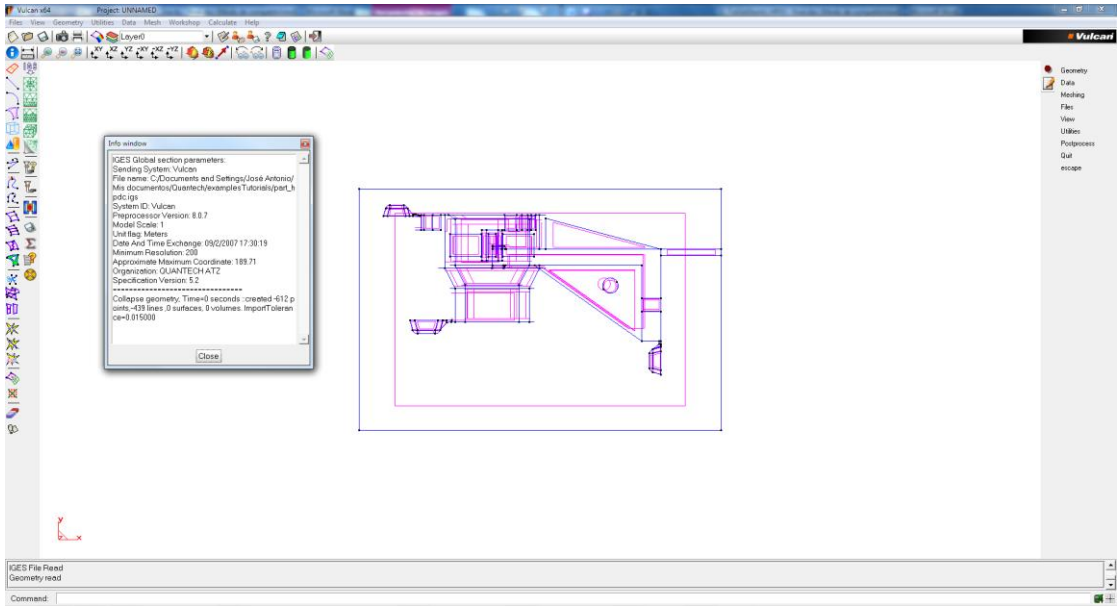


Figure G.5. IGES read.

Close the info window and have a look at the geometry of ***Figure G.5*** by right-clicking and doing **rotate-trackball** and **zoom in** and **out**.

Once done with looking at geometry, we will start working with different layers: one layer for the casting, one layer for the mould, etc. Click on the Layers icon:

The following window will appear (***Figure G.6***):

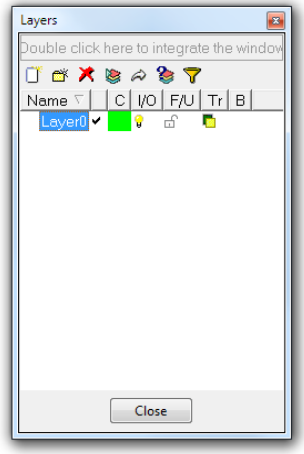


Figure G.6. Layers control window.

Perform the following operations:

- Click on **New** button, and create a new layer.

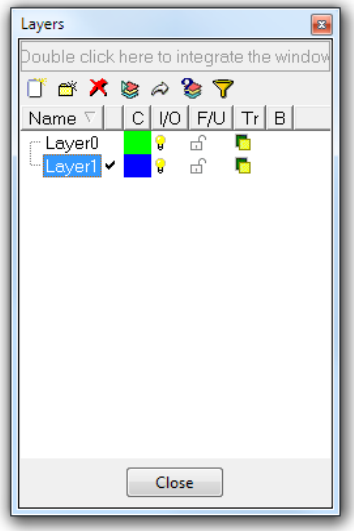


Figure G.7. Layer selection and layer to use.([3])

- **Select** the layer Layer0, enter the name “cast” in the textbox and click on **Rename** button.
- **Select** the layer Layer1, enter the name “mould” in the textbox and click on **Rename** button.

NOTE ([3]): The **selected** layer is highlighted in black. After selecting a layer by simply clicking on it, we can delete it, rename it, change its color, turn it on, and off, send elements to it, etc. The **layer to use**, instead, is the layer with the checkmark: in the layer to use we can create geometry, delete it, etc. in the drawing. Summary: **selected** is to make changes using layers window, and **layer to use** is the layer used to create, delete, or modify geometry. We can change **the layer to use** by selecting a layer and click on **Layer to use** icon.

- Be sure that **Layer to use** is cast; it says so beside the button, and there is a checkmark (✓) in the layer name.
- Select the layer mould, and click on **Send To** button, and then Surfaces.
- Go to the geometry and select the six surfaces that form the exterior of the mould.
- Turn **off** the layer mould.
- Select the layer cast, click on **Send To** surfaces again, and send all the casting surfaces to the layer Cast.
- **Close** the layers window.

The resulting geometry at this moment shall look like ***Figure G.8***.

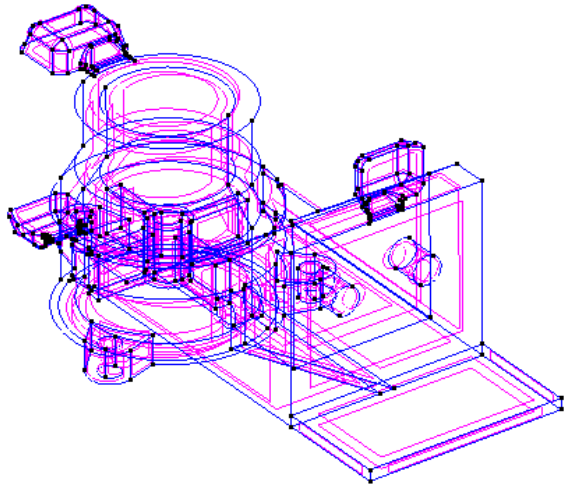


Figure G.8 High pressure die casting geometry.

The geometrical entities in Vulcan have the following hierarchy order:

Points → Lines → Surfaces → Volumes

We can interpret the hierarchy by higher entities and lower entities. For example, a surface is higher than a line, a surface is lower than a volume, etc; we cannot delete a surface if there are related higher entities present (Volumes).

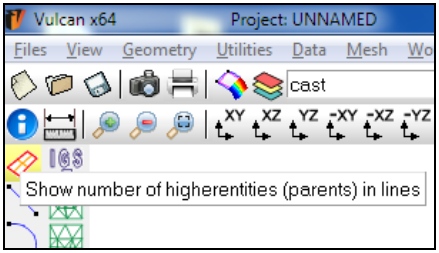
In ***Figure G.8*** we can see the points in black, lines in blue and surfaces in pink. There are always points that define line ends, lines that define surfaces, etc.

Now that we have a geometry to work with, let's begin with the next step, geometry treatment.

Step 2

GEOMETRY TREATMENT

This geometry apparently has no errors, so we won't collapse points or lines. Let's check the integrity of the geometry by selecting **Draw higher** option.


Vulcan x64
Project: UNNAMED
Files View Geometry Utilities Data Mesh Wo
cast
XY XZ YZ -XY -XZ -YZ
Show number of higherentities (parents) in lines

The result shall look like this:

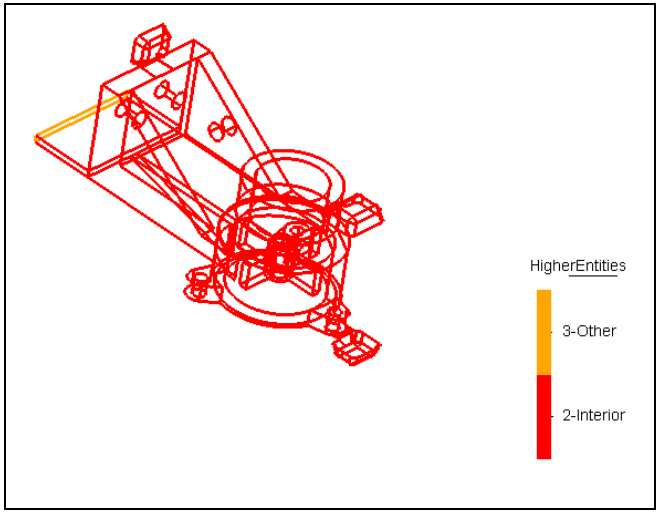


Figure G.9. Draw Higher screen.

In ***Figure G.*** we can check the geometrical integrity of the model before we mesh it. To form a closed surface, all the lines need to be the end of two surfaces (“Interior” in the figure). If there would be open edges on the geometry, the lines would appear under a different colour and labelled “Boundary”, or “Isolated” lines.

The lines at the entrance of the material (“Other”) are connected to more than two surfaces because they are also connected to the mould surfaces, so they have a different colour; this is correct. The other lines are all “interior” (in red). This means that all the lines are connected to two surfaces (the lines are not forming open boundaries or are isolated lines). In other words, this geometry, consisting of points, lines and surfaces, is sound.

We can rotate, and zoom the casting while we are in **draw higher** mode.

Now let’s press **Esc** to go back to the geometry.

We have now a repaired geometry. Let’s move on to the next step.

Step 3

MESH GENERATION

Before start the meshing generation, we will set the Meshing parameters as shown in ***Figure G.5***.Go to **Utilities** → **Preferences** and select the label **Meshing**. Set all the parameters as shown in the figure:

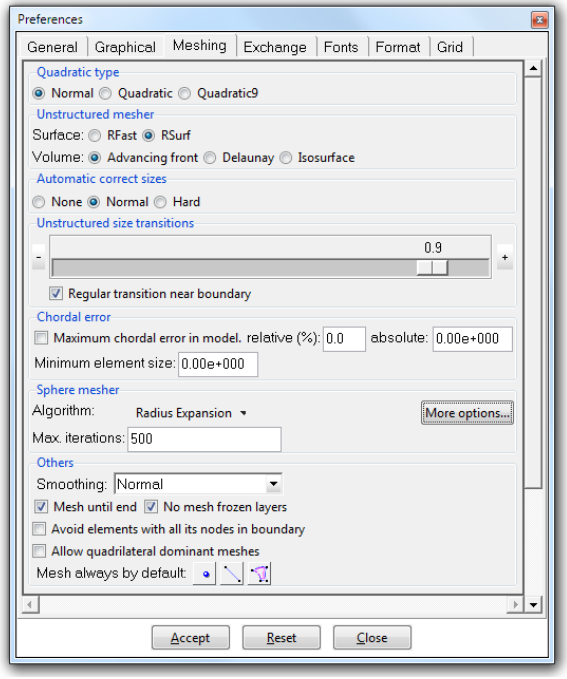


Figure G.5. Meshing preferences.

Click **Accept** and then **Close** the window.

It is recommendable to first generate a (triangular) surface mesh in order to generate a good (tetrahedral) 3D mesh, because this way we can check the mesh quality faster. The steps to generate a good tetrahedral mesh will be:

- Generating a surface mesh
- Check the quality of the mesh generated and make corrections, if necessary
- Return to geometry view and generate a geometrical volume out of the casting surfaces
- Generate a tetrahedral mesh of the casting, overwriting the previous mesh
- Check again for mesh quality
- Return to geometry and generate the mould volume, and then generate the final casting and mould mesh, overwriting the previous mesh; this is the only mesh needed; the previous ones were generated for verification

Save the project before meshing. Go to **File → Save as…**

Go to the Mesh menu as shown in ***Figure G.6***:

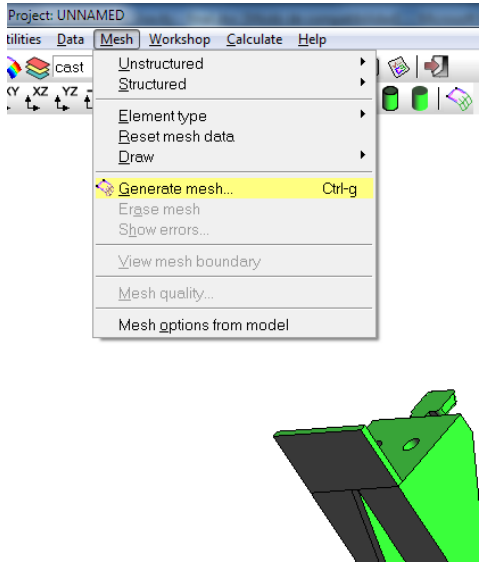
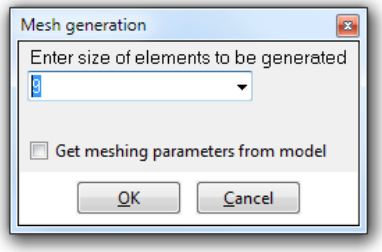


Figure G.6. Mesh generation screen.

The automatic assigned size of the mesh is **20.5**; change this value to **9:**



and click **OK**. The result will be like this: ([4])

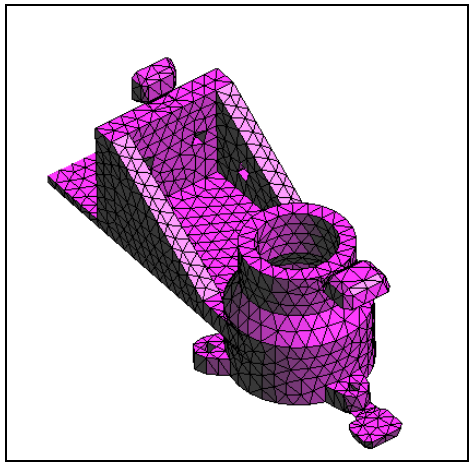


Figure G.7. Non-uniform triangle mesh.

NOTE ([4])*: We can toggle between geometry view and mesh view by clicking on the icon:* *and also change normal view to render view by right-clicking on the geometry and selecting* ***Render →Flat*** *or* ***Render →Normal***

We can see in ***Figure G.7*** that elements are not too uniform in order to be acceptable. Let's decrease the size of the elements to obtain a more uniform mesh.

Go to Mesh → Generate Mesh…
And set a mesh size of 5. The result should be like in ***Figure G.8***:

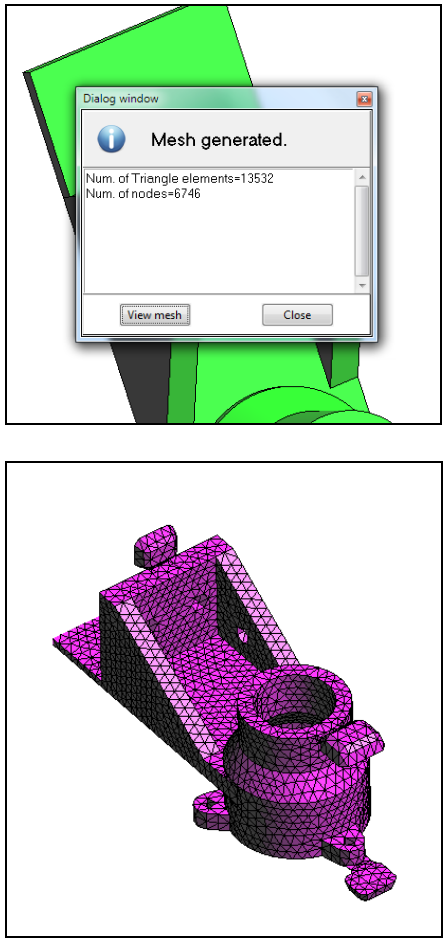


Figure G.83. Mesh generation.

Compare the results of ***Figure G.7*** and ***Figure G.8***, to see that, in order to improve the mesh quality, we have to decrease the elements size. The mesh in ***Figure G.8*** is a more uniform mesh, but the number of elements had increased.

Let's now return to geometry by clicking and generate the volume and tetrahedral elements.
Go to Geometry → Create → Volume → By contour or click the **Create volume** icon .

The following will appear in the command line:
"Enter surfaces to define volume (ESC to leave)"

Select all the surfaces of the casting with a selection window ([5]):
"Added 173 new surfaces to the selection. Enter more surfaces. (ESC to leave)"
press **Esc**

"Created 1 new volume. Enter more volumes"
And now press **Esc** again:
"Leaving volume generation"

Now regenerate a mesh with size of **5** in order to obtain a mesh of the volume. The result will look again as in ***Figure G.8***.

Generating the mould mesh

Return to geometry view. Click on Layers icon, and turn on the layer **mould**. With the layer mould selected, double click on the layer mould.

Now, we will generate a volume for the mould.

Go to **Create volume**, and select all the surfaces with a selection window:

NOTE ([5]): We can use the Layers icon to facilitate the selection of the casting surfaces. Leave the **cast** layer on and the **mould** layer off for this selection.

Rotate the geometry in order to see the material entrance slot:

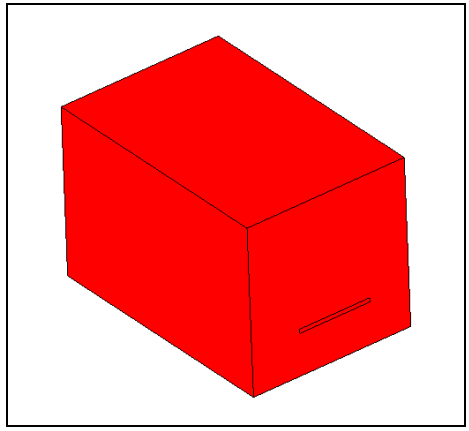


Figure G.9. Mould volume generation.

Now zoom on the slot (material entrance) and click on the small surface in order to deselect it. The result should look like this:

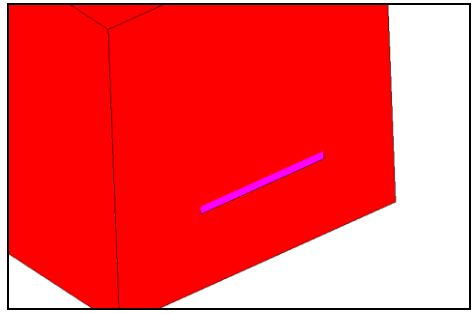


Figure G.10. Mould volume generation.

Now press **Esc** twice and the mould volume will be generated.

Assigning mesh size to the mould

In the case of the mould geometry, we will try to use tetrahedral elements as big as possible, in order to reduce the total number of finite elements of the problem. This will reduce the calculation time; so, instead of assigning an automatic size of 5 to all the geometry (cast + mould) we are going to do the following:

Turn off the layer **cast**.

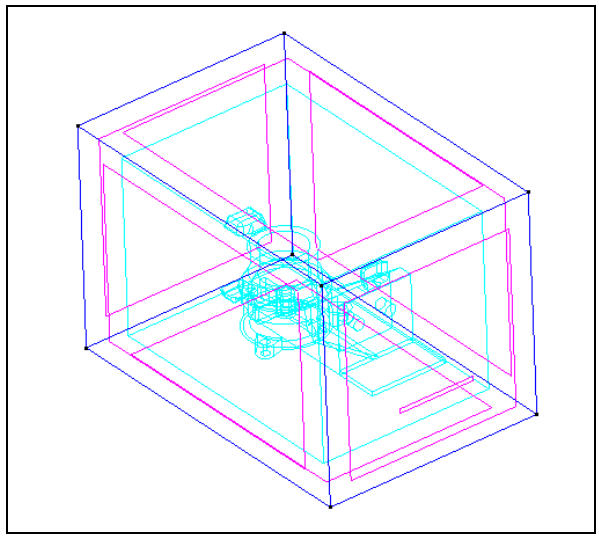
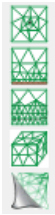


Figure G.11. Mould layer.

And now, we will see in detail how to assign properties to the mesh.

Using the **assign mesh sizes** icon, we will assign properties to the geometrical entities of our geometry:



We have to assign mesh sizes in geometrical hierarchy order: **Points, Lines, Surfaces and Volumes.**

Let's assign a mesh size of **30.0** to all the entities of the mould:

a) Click assign sizes to points, the first icon .

The following window will open:

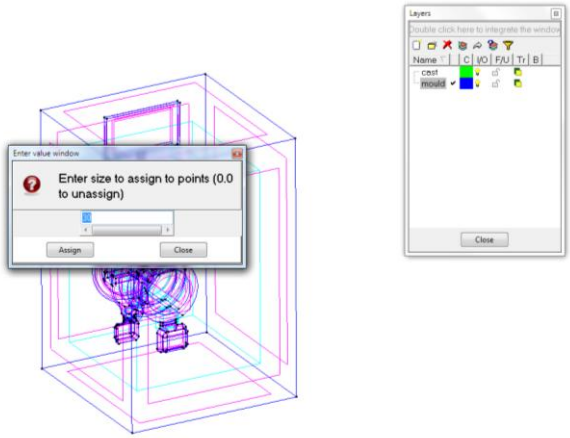


Figure G.12. Mesh generation

b) Assign a size of **30** in the window and click the **Assign** button.

c) Select all the points of the mould, with a selection window:

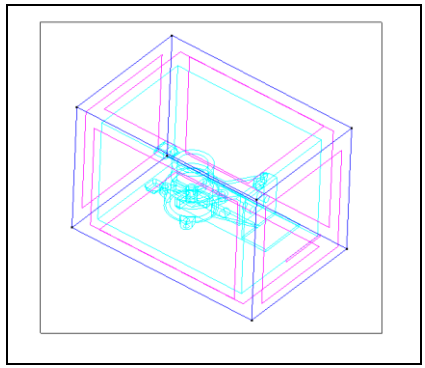


Figure G.13. Selection window

d) In the command line, the following message will be displayed:
"Added 8 new points to the selection. Enter more points. (ESC to leave)"

These are all the points added to the selection; the points are added, and as it says in the command line, we have to press **Esc** to accept the selection and leave.
The following message will appear in the command line:

"Assigned size=30 to 8 new entities"

e) click **Close** to close the enter value window.

Now we repeat **points a) to e)** for lines, surfaces and volumes, with the same size of 30.

Also we can use the button to assign directly the size to the points, lines and surfaces.

Once the sizes are all assigned, we generate the mesh again (same general mesh size: 5). The result shall look like this (***Figure G.19***):

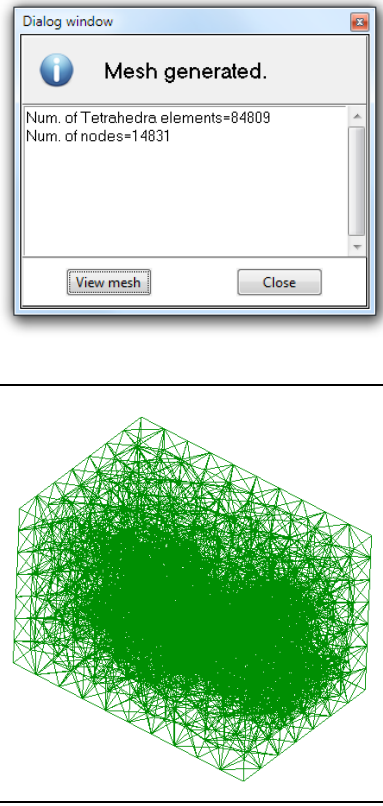


Figure G.19. Generated mesh

Now that the mesh is generated, we will see in detail how to set the process parameters for this particular type of problem, high pressure die casting.

We will continue the step-by-step process with the **Process bar**.

Step 4

FOUNDRY PROCESS DEFINTION

Until now we were dealing with geometrical parameters, definitions, etc.

From now on, we will deal with real foundry parameters, characteristics, processes, etc. It is very important to set the parameters properly. To have closer parameters to reality, give us more accurate results. So read on and let's do casting simulation.

The first step we take once we have the casting and mould geometry defined and meshed, is let Vulcan know what type of process we are going to calculate. Click on **define the main process characteristic**, the 7th icon:

A window with 3 labels will appear. In the **Process type** label, select **High pressure,** as shown in ***Figure G.14***

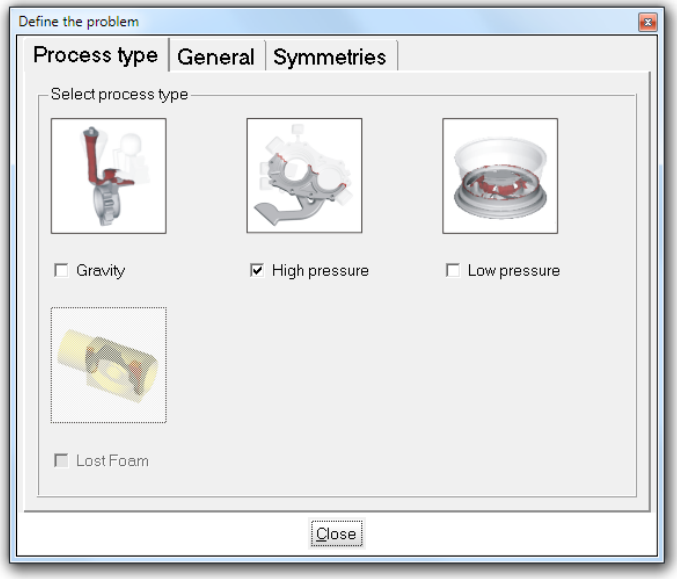


Figure G.14. The Process type window

Once this step is taken, Vulcan adjusts all the internal parameters and the remaining icons to this particular type of process.

In the **General** label, we have a set options to define: units, gravity direction (with respect to the coordinate system compass), environment temperature, etc.

Set the gravity direction to +Y, and the Environment temperature to 25 Degrees Celsius, as shown in ***Figure G.15***:

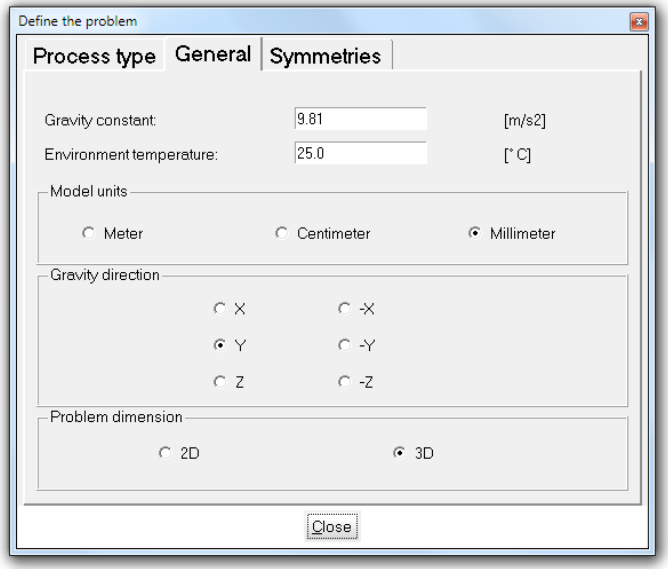


Figure G.15. The Process type – General window

In the **Symmetries** label (***Figure G.16***) we have to define the symmetry planes created; if there are no planes of symmetry, simply leave No planes selected:

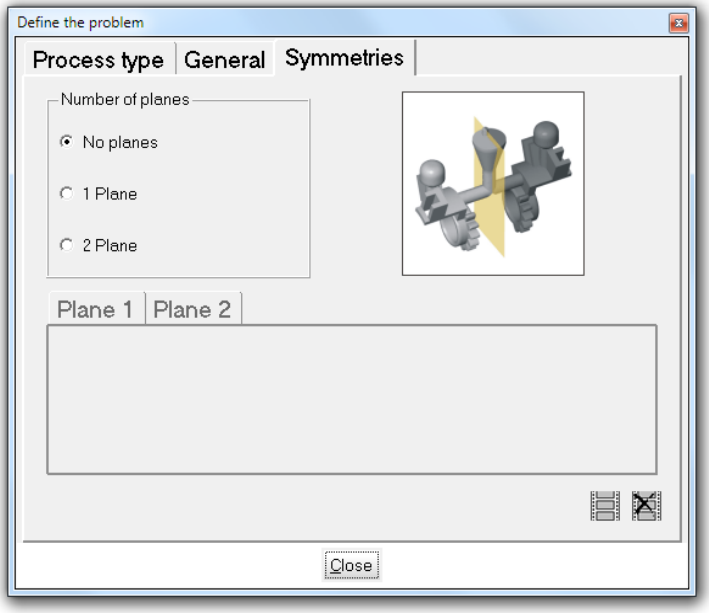


Figure G.16. The Process type – Symmetries window

Now the gravity casting process is defined; click **Close** ([6]) to continue to the next step.

NOTE ([6])*: All the changes made will be saved upon clicking on the* ***Close*** *button.*

Step 5

FOUNDRY COMPONENTS DEFINTION

Click on the **Components definition** icon:

The following window will open (***Figure G.17***):

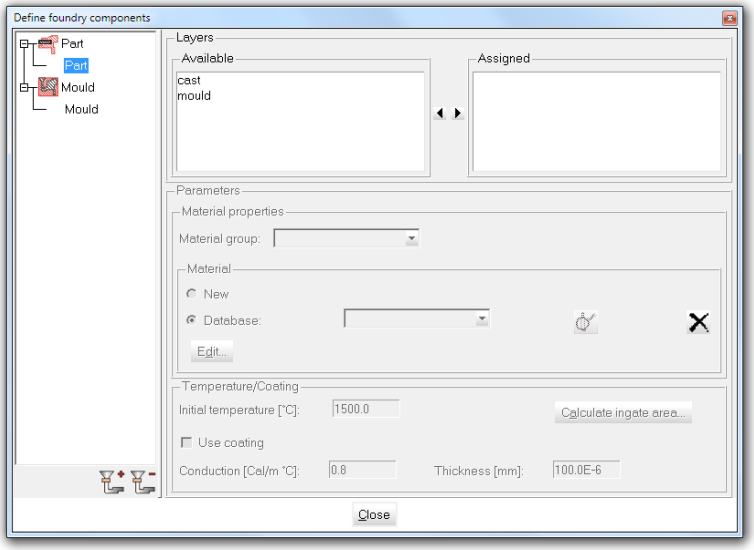


Figure G.17. Foundry components definition.

First we have to assign each layer to the corresponding foundry component. On the example shown, the Part component is highlighted on the tree. In the **Available** window, select **cast** and click on the Icon. The **cast** layer will move to the **Assigned** window.

Now, select the Material group and specific material from the database, and set a temperature of 650 Degrees Celsius, as shown in ***Figure G.18***:

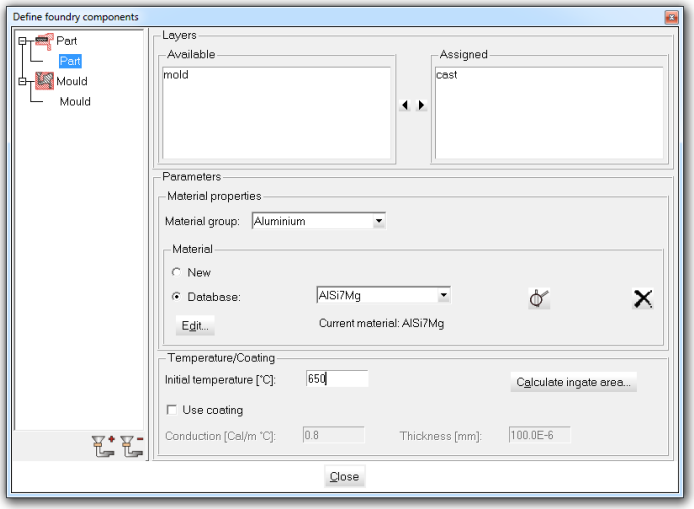


Figure G.18. Foundry components definition.

Now perform the same procedure for the mould, first selecting the **Mould** component on the tree, and then **mould** from **Available** window. Again, it should move to **Assigned** window.

Assign in the **Mould materials** group, the **Steel** material at initial temperature of 300 Degrees Celsius:

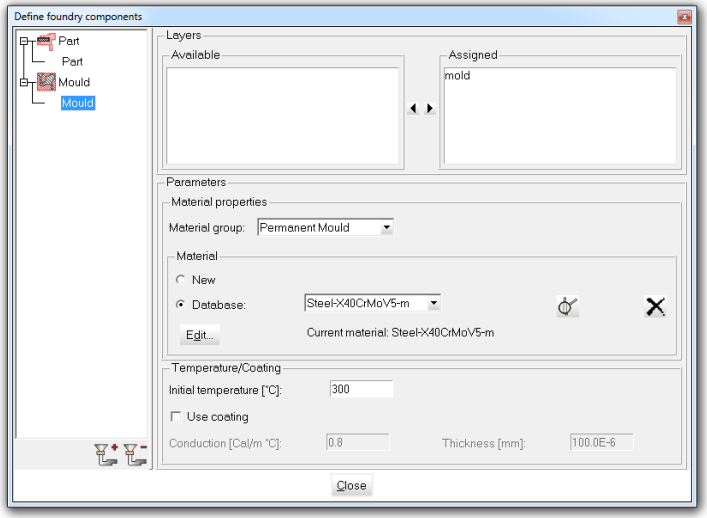


Figure G.19. Foundry components definition.

Click **Close** to save the changes.

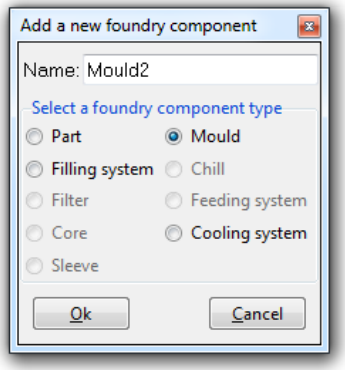
NOTE: Let's explain some of the icons that appear in the **Components definition** window. You can skip this note to continue with the example, and go to the .Step 6

Click again on the icon:

In case of having multiple layers, you can assign and unassign layers to foundry components by using the *icons (for example, in case of having the mould divided in upper part and lower part on different layers, or in case of having to make changes).*

Now click on the ***Add new foundry component*** *icon:*

The following window will appear:



This window has to be used in case of need to insert more components to the tree, such as more parts of the mould, or cores. On the example shown, there are only two layers, the whole mould and the part, so there is no need to create a new foundry component.

*In **Figure G.19** we can also see the **Use coating** checkbox. If we use this option, we can change the conductivity and the thickness for the specific coating we are using.*

*Now click on the **Edit…** button; this window will open:*

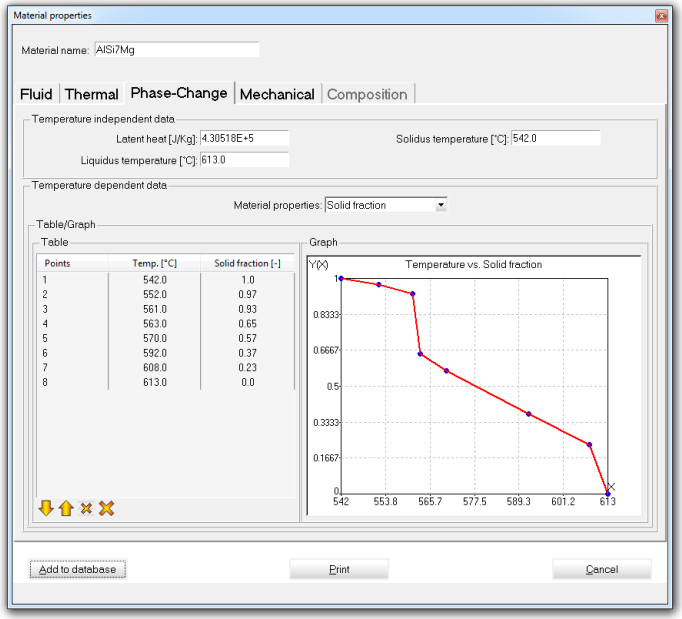


Figure G.20. Editing material properties

*Here we can see the material properties in both table and graphical views. We can change these data, add new materials, etc. See the Tutorials for more details. Now click **Cancel** on this window.*

*And click **Cancel** to continue with the example.*

Step 6

OPERATIONS DEFINITION

Now we are going to define the specific operations to be simulated. Let's click on the **Operations** icon:

The following window will appear (***Figure G.21***):

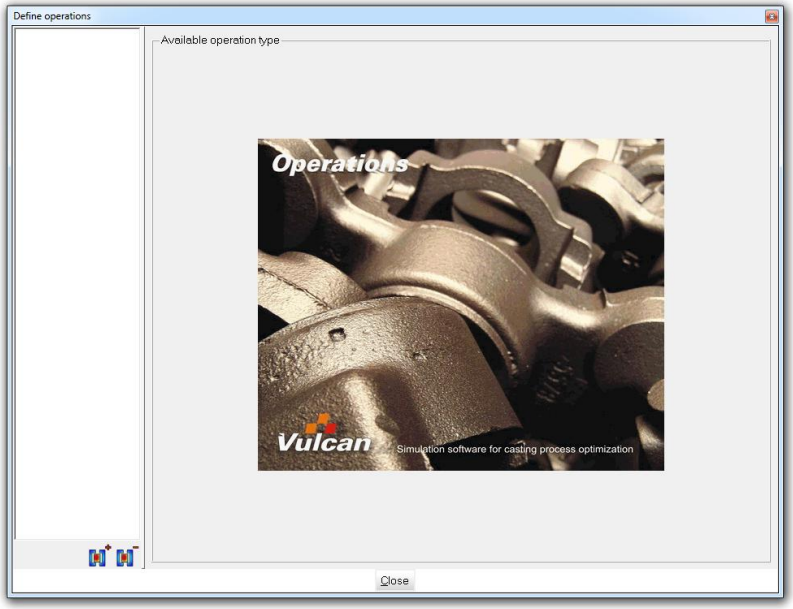


Figure G.21. Operations definition.

From the point of view of casting simulation, we can divide the casting process into three parts: **filling**, **solidification**, and **cooling**.

The filling begins when the material starts to enter into the mould, and finishes when the molten metal fills the entire mould.

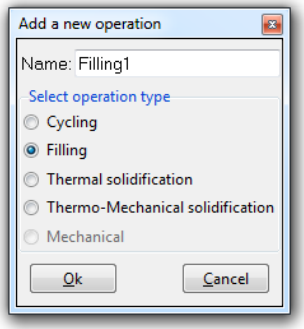
At this moment, solidification starts, as metal begins to cool down and solidify; solidification finishes when the last part of the casting turns into solid phase.

At this moment, cooling starts, and goes all the way down until the casting and mould reaches room temperature.

In our gravity casting example, we are going to run a simulation the **filling** and **solidification** parts of the process.

Click on the add operations icon:

The following window will appear:



Here we have to select all the operations we want to simulate one by one:
Leave Filling and click OK, and then enter on the same previous icon, and select **Thermal solidification** operation.

The result should look like this:

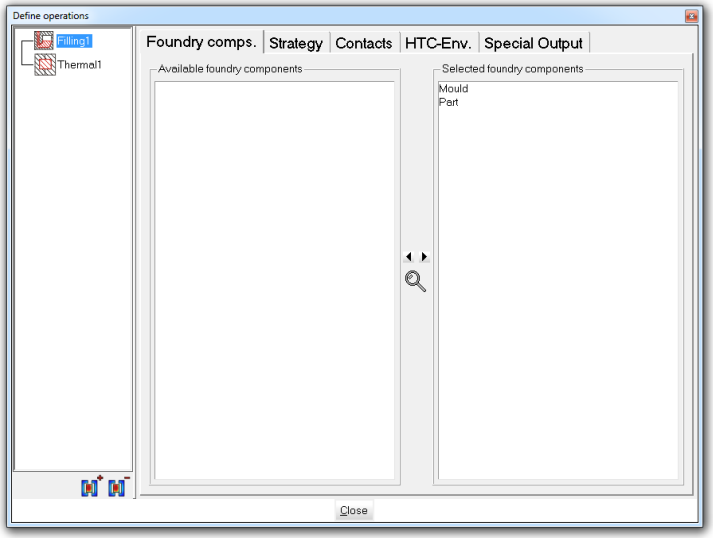


Figure G.22. Operations definition.

In ***Figure G.22*** we can see that all the foundry components are assigned by default to the filling and solidification operations. In (***Figure G.17***) we created those foundry components. Here we have the option of changing the foundry components, selecting different components for different operations, etc.

To continue with the example, we leave the default selection, and then, we are going to view and set all the parameters for both operations. We have to go trough all the labels and change the default parameters for those of interest. In our example, we will see all the options and a short explanation of the main parameters.

In strategy label (***Figure G.29***), we set the piston diameter, and we build the Time vs. Velocity curve, for the specific injection machine. On the example, we set a piston diameter of 70 mm, and change the piston velocity to 2.5 m/s, for times 0 and 0.5 seconds, as shown in figure:

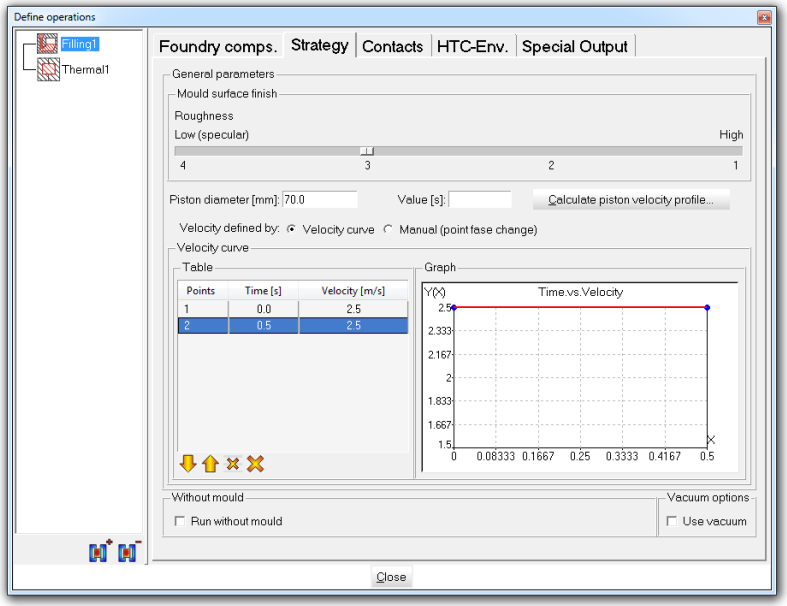


Figure G.29. Strategy label

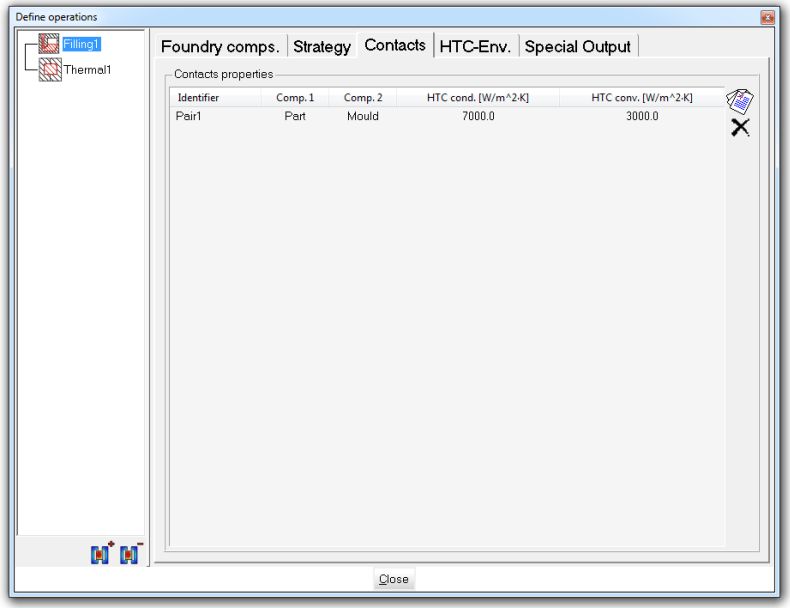


Figure G.23. Contacts label

In the Contacts label (***Figure G.23***), we set the values of the Heat Transfer Coefficients (HTC) for the contact between the casting and the mould. This contact is characterized by two different values: HTC conduction and HTC convection. Change values to:

HTC conduction: 2000
HTC convection: 1000

And continue to the next label.

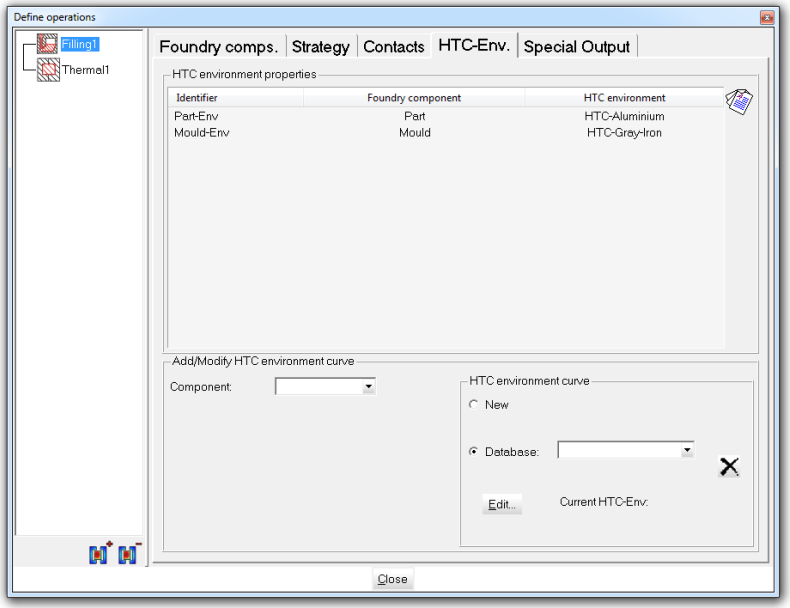


Figure G.24.HTC-Environment label

Let's have a look on ***Figure G.24***. Here we set the value of heat transfer coefficient with environment (HTC-Env) in this case, environment is the air at the temperature set on the problem definition ([7]).

To continue with the example, leave all the default options and continue to the next label.

NOTE ([7]): We can modify or create a new value for this parameter by entering on the Edit... button and setting a new value. The parameter could change, for example, if the air is not still (there is air circulation with fans)

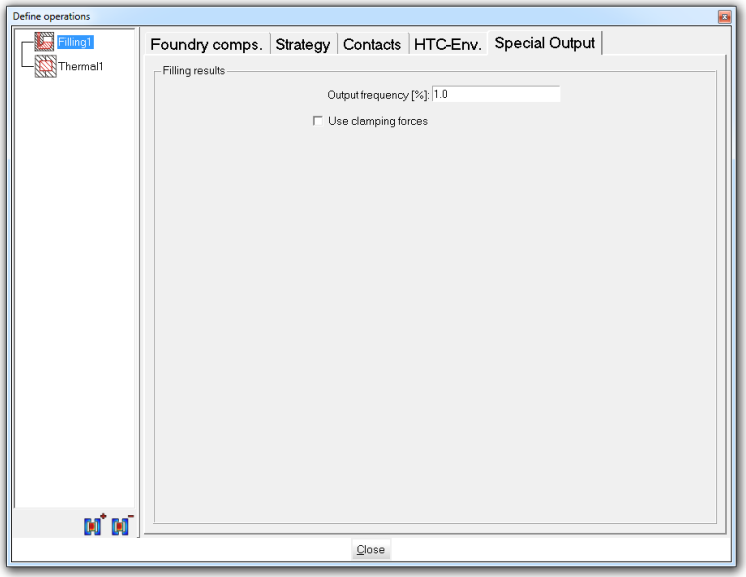


Figure G.25. Special Output label

In ***Figure G.25***, we define the number of results to be written during the analysis. Leave the default option.

In HPDC we can select the clamping forces of the mould during the filling.

Now, change to **Thermal1** operation on the operations tree, as shown in ***Figure G.26***:

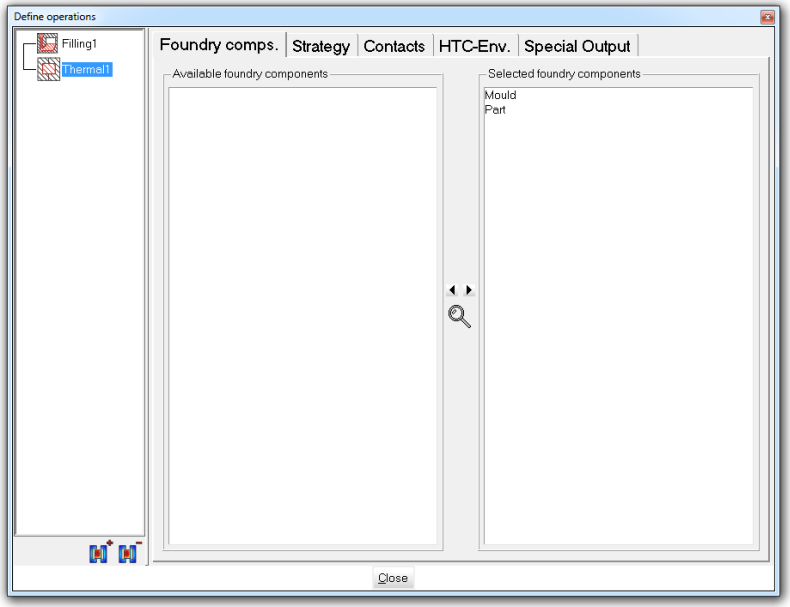


Figure G.26 ***Thermal1*** *settings*

In this screen, foundry components are already selected. Go to the next screen:

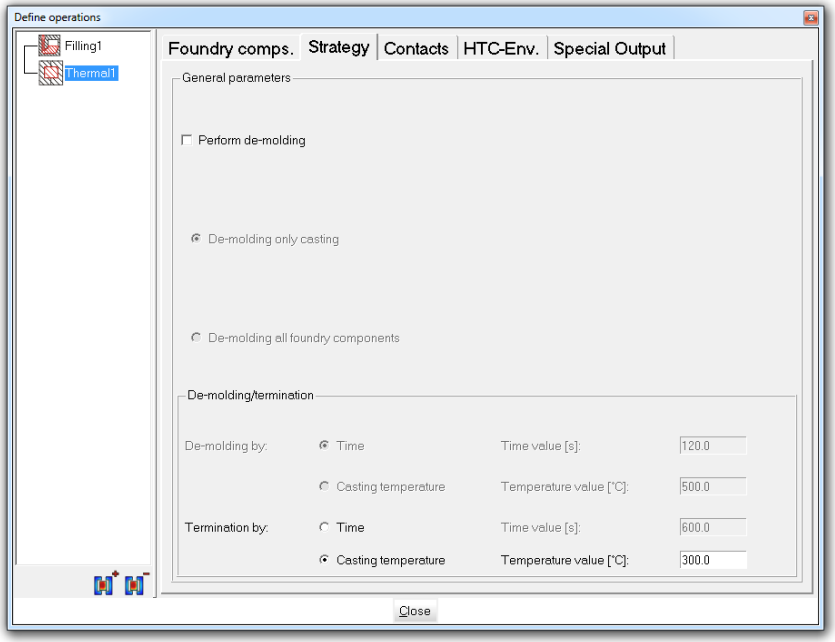


Figure G.27. Thermal Strategy label

In this screen, we define the **De-moulding** and **Termination** characteristics of our process. In the current problem, we leave de-moulding deselected, and we select termination by Casting temperature, 300 ºC. Click on the next label.

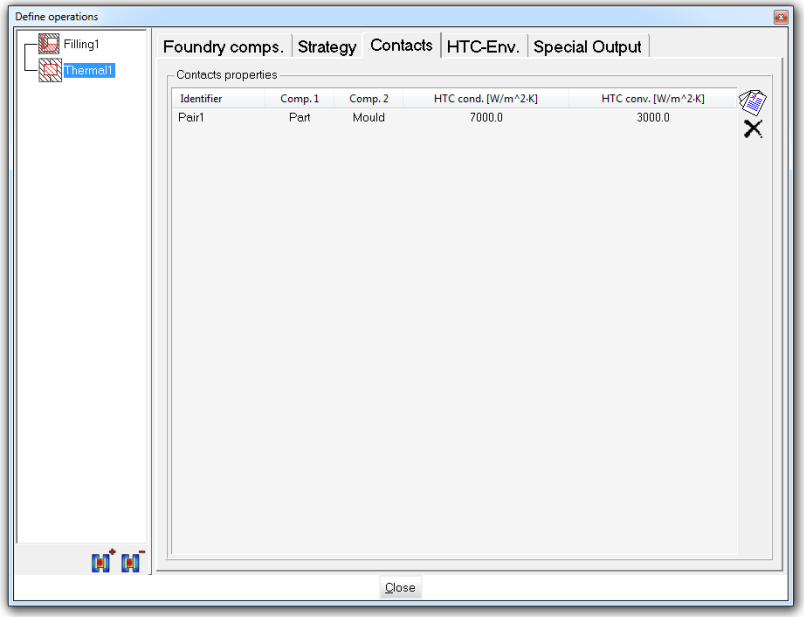


Figure G.28.Thermal Contacts label

Here again we leave the HTC conduction in 2000, and HTC convection in 1000, and continue to the next label.

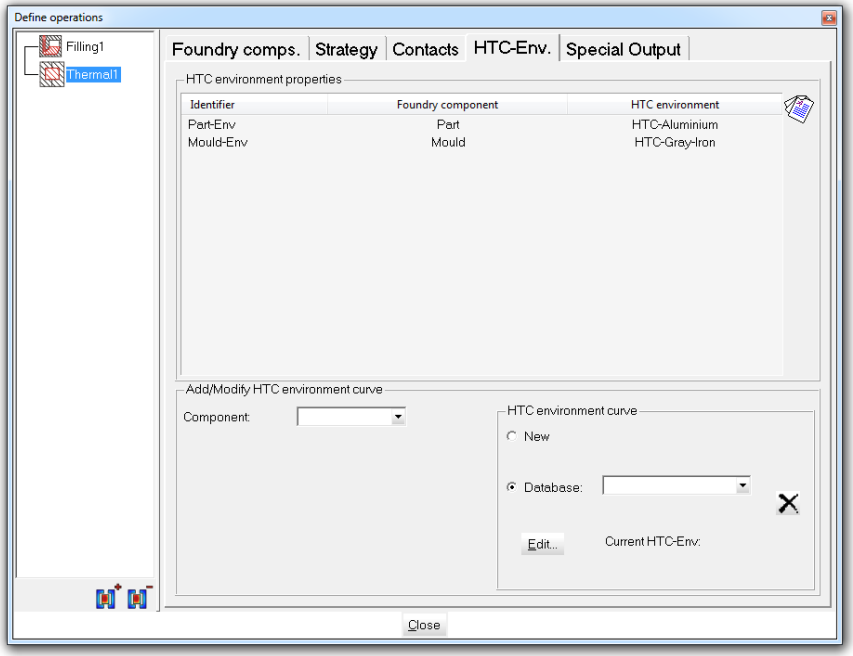


Figure G.29.HTC-Environment label

We have pre-selected the same options as for the filling analysis. Continue to the next label.

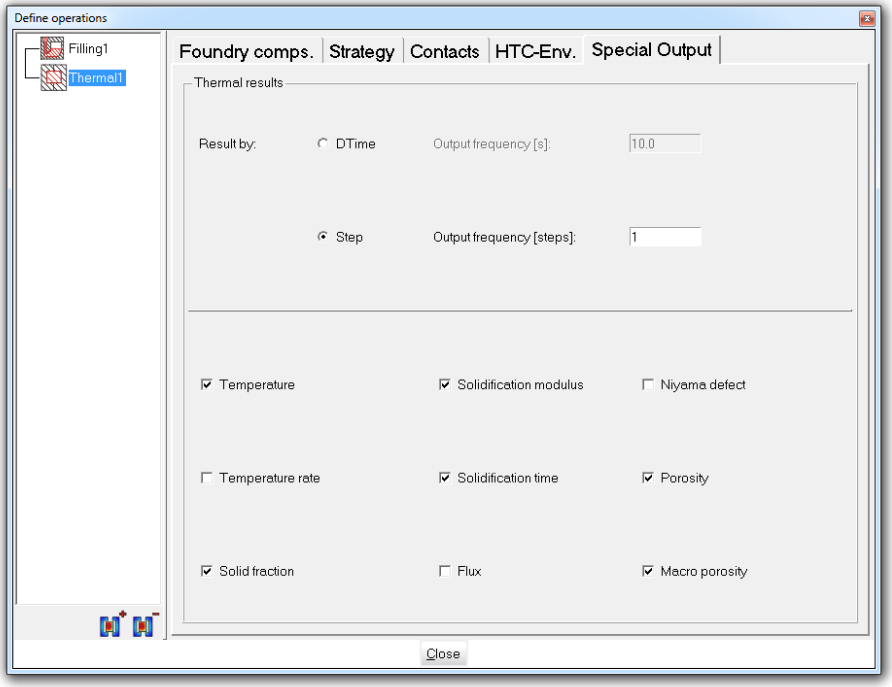


Figure G.30.Thermal Special Output label

In this label, we define the thermal results that we are going to visualize on the post-process. In the example, leave Temperature, Solid fraction, Solidification modulus, and Solidification time.

Now that we have defined all the problem parameters, click on the **Close** button. The parameters will be saved.

Step 7

SAVE PROJECT

Before start the calculation, don't forget to save the project:

Click on **Save** icon.

NOTE: the filename must not contain spaces or symbols.

Step 8

CALCULATION

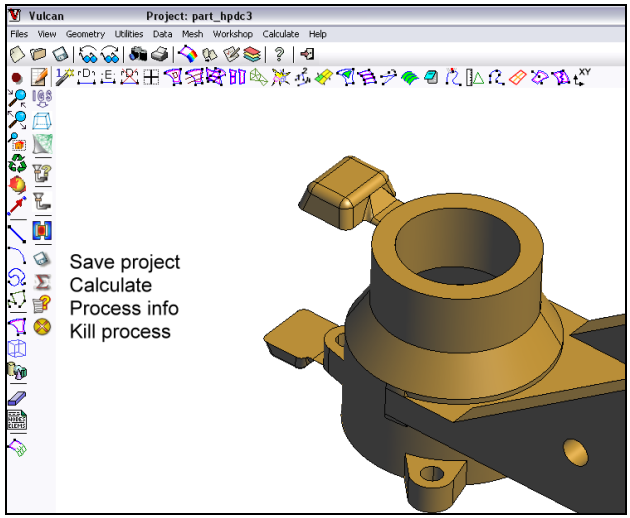


Figure G.31.Calculate, Output view and Terminate Process icons.

Now, following the column of icons shown in ***Figure G.31***, we first saved the project and then **Calculate** it. The calculation time will depend upon the complexity of the problem.
The calculation window will look like this:

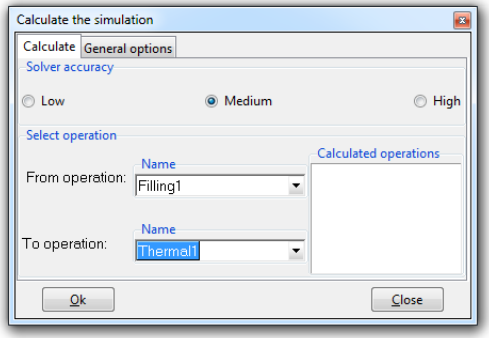


Figure G.32.Calculation window

Select **From operation** filling **To operation** Thermal1 and click OK. The calculation will start:

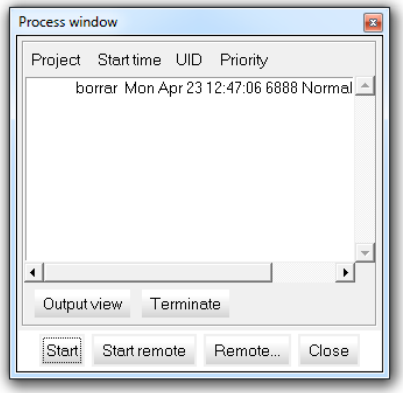


Figure G.33.Calculation window

We can view the **Output** view by clicking the icon and see the calculation status, and also interrupt and cancel the calculation by **Terminate** the process, make changes and **Calculate** again. Close the window.

OUTPUT VIEW

Once the window is closed, we can still access to the output view by clicking on the icon:

And we will see the calculation process:

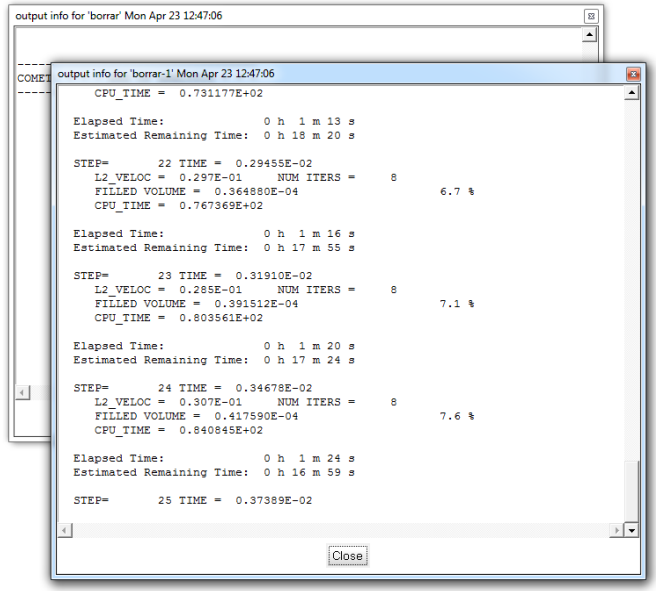

output info for 'borrar' Mon Apr 23 12:47:06
output info for 'borrar-1' Mon Apr 23 12:47:06
CPU_TIME = 0.731177E+02
Elapsed Time: 0 h 1 m 13 s
Estimated Remaining Time: 0 h 18 m 20 s
STEP= 22 TIME = 0.29455E-02
L2_VELOC = 0.297E-01 NUM ITERS = 8
FILLED VOLUME = 0.364880E-04 6.7 %
CPU_TIME = 0.767369E+02
Elapsed Time: 0 h 1 m 16 s
Estimated Remaining Time: 0 h 17 m 55 s
STEP= 23 TIME = 0.31910E-02
L2_VELOC = 0.285E-01 NUM ITERS = 8
FILLED VOLUME = 0.391512E-04 7.1 %
CPU_TIME = 0.803561E+02
Elapsed Time: 0 h 1 m 20 s
Estimated Remaining Time: 0 h 17 m 24 s
STEP= 24 TIME = 0.34678E-02
L2_VELOC = 0.307E-01 NUM ITERS = 8
FILLED VOLUME = 0.417590E-04 7.6 %
CPU_TIME = 0.840845E+02
Elapsed Time: 0 h 1 m 24 s
Estimated Remaining Time: 0 h 16 m 59 s
STEP= 25 TIME = 0.37389E-02
Close

Figure G.34.Output view.

TERMINATE PROCESS

If we want to change parameters and calculate again, we would click on the last icon:

This icon will terminate the process. In the example, we will leave the computer calculating and wait for the end of calculation. We will not click on the icon.

Exiting Vulcan while is still calculating

When the pre-process is over and we are calculating a process, we can exit Vulcan application but without interrupting the calculation:

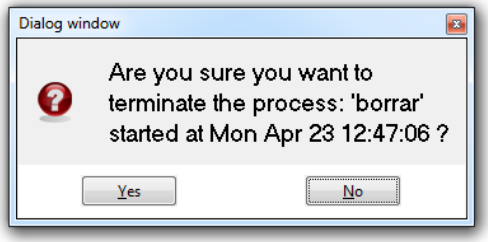


Figure G.35. Exiting Vulcan while is still calculating.

Click **No** and Vulcan will exit. We can open Vulcan later, during or after the calculation. The computer, of course, must remain on.

Step 9

POST-PROCESS

Post-process means the visualization of the simulation results; this is perhaps the most important step of all the casting simulation steps; here is where we read and analyze the simulation results.

Once the calculation is finished, we can visualize the results of this simulation. There are many different ways to do it, like colour maps, vectors, iso-surface curves, virtual thermocouples, etc.

Figure G.36 shows the window that appears once calculation step is finished; we can directly access to post-process by clicking on **Postprocess** button.

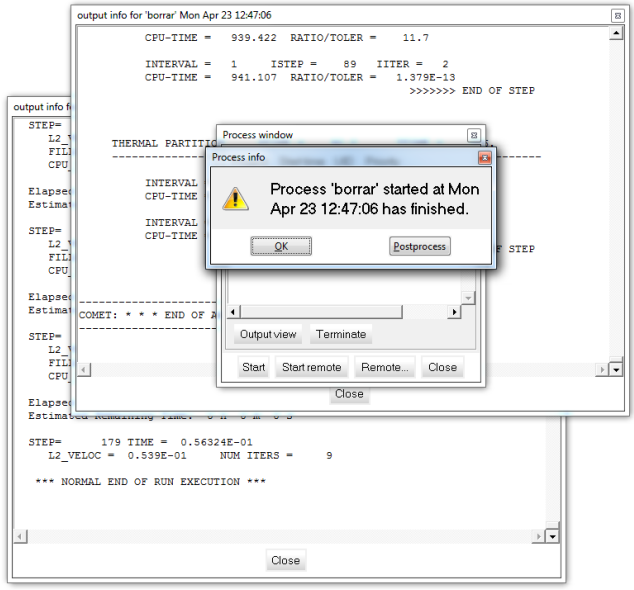


Figure G.36.Process info.

Another way to switch between pre- and post-process is by clicking the **toggle** ([8]) icon .

The first step in results visualization is to load results of any **foundry operation** (Filling, thermal solidification, thermo-mechanical solidification). Let's load the filling operation and analyze the results.

Go to **Vulcan results** → **Load results**…

And select the **Filling1** operation:

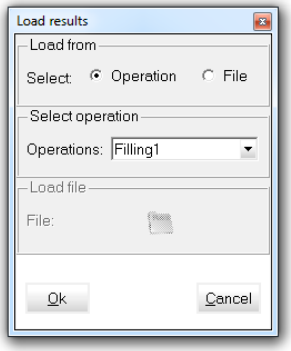


Figure G.37. Results loading

Click **Ok** and the filling results will be loaded.

NOTE ([8]): Every time we toggle, when we return to post-process, we have to load the results.

Now that we have all the filling results loaded, let's begin, for example, to see the evolution of the casting material in the mould; where there could be air entrapment; or the advancing fronts of material could encounter; the last spots of the mould to be filled, etc.

To see this, we will use a tool called filling mater. In this animation, we will see the material free surface in red, the filled areas in grey, and the mould will be transparent.

Go to main menu:

Vulcan results → **Filling mater** or select the button

Once the Animate window appears, press the **Play** button:

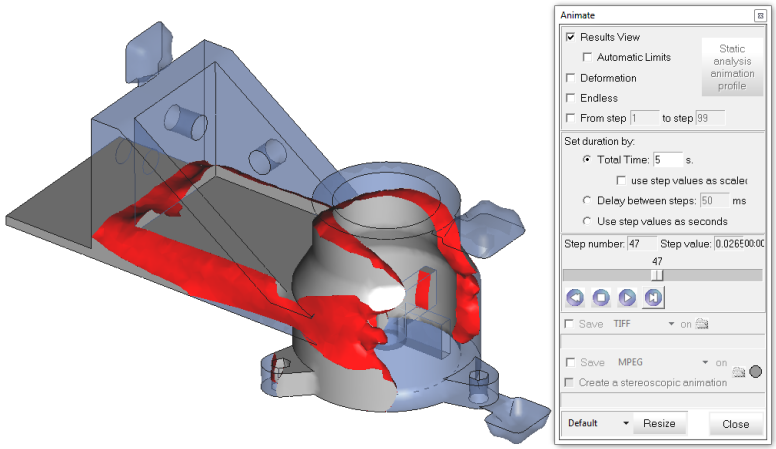


Figure G.38. Filling mater animation.

In ***Figure G.38*** we can see the **Animate** window and the control buttons. We can keep this window open while we rotate, zoom in and zoom out the casting. Let's explain the basic Animate controls.

Basic Animate window controls

Delay: We can set this time in order to “slow down” the animation; delay time is not the real time, but a time we can increase in order to ease the visualization.

Step: This box shows the real time step. It shows the time in seconds since the filling process had begun.

Then we have the animation controls:

these controls are the standard Play/Pause/Stop/Fast Forward/Rewind controls. We can control the evolution of the animation with them.

Let's say now that we want to see other phenomena, for example, turbulences formation, highest filling velocities inside the casting, velocities at which the advancing fronts encounter, etc. We will use a different representation to see these results.

Go to:

Vulcan results → **Filling vectors** or select the button

And again, the animate window will show up. The animation will show the velocity vectors inside the casting:

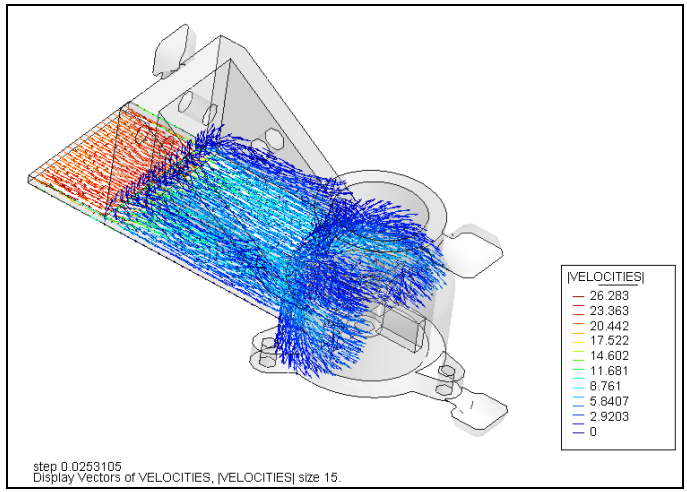


Figure G.39. Filling vectors animation

The colour scale represents the module of velocity. In ***Figure G.39*** we can see the highest velocities in red, and the lowest velocities in blue.

Now we want to see parameters like temperature of front encounters, early cooling during filling, and possible solidification during filling.

To see this kind of results, we will go to:

Vulcan results → **Temperature evolution** or select the button

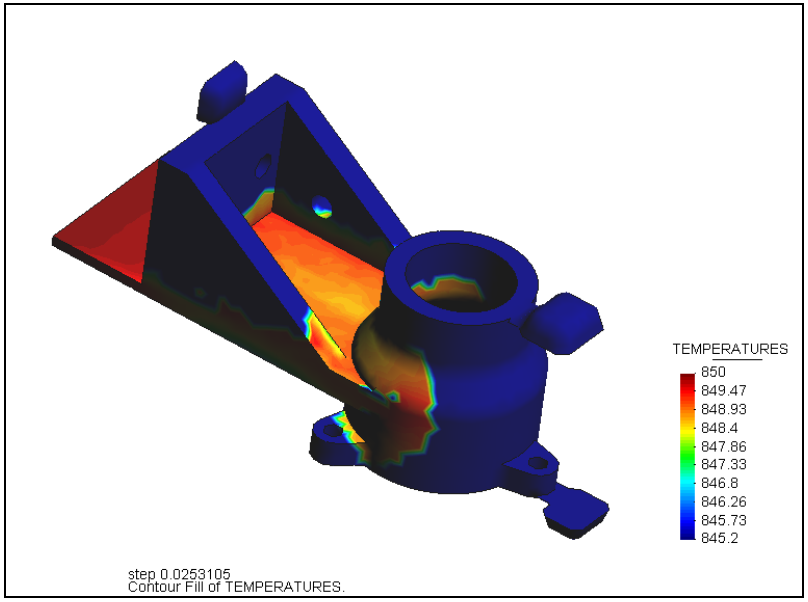


Figure G.40. Temperature evolution during filling.

Here we can see the temperatures and the temperatures colour scale, and also the time step since the start of filling.

Now we will load the second **foundry operation** we had run: solidification.

Solidification goes from the end of filling until the casting is completely solid.

Let's see the results. Go to:

Vulcan results → **Load results…** and load the **Thermal1** operation:

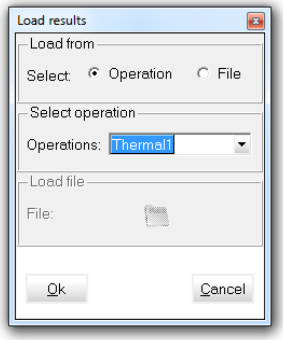


Figure G.41. Thermal operation loading.

Once the results are loaded, click Ok and let's see some solidification results.

This time we will see temperature evolution in both casting and mould, excess temperature on the chillers, cores, etc.

Go to:

Vulcan results → **Temperature evolution** or select the button

The result will be like this:

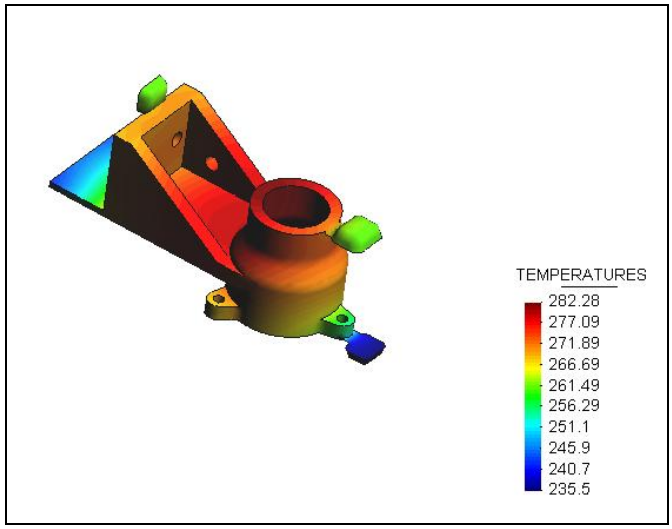


Figure G.49. Temperature evolution.

In this result, we can animate and see the temperature evolution in the casting surface, but it would be interesting to "cut" the casting and see the temperature inside.

To do so, we will make a cut of the casting and mould.

Go to:

Vulcan results → **Results evolution with graphs**… or select the button

Select **Cut xz plane** button, and then draw a line dividing the mould horizontally in two equal halves. Selecting the Show cut sections, the result will be:

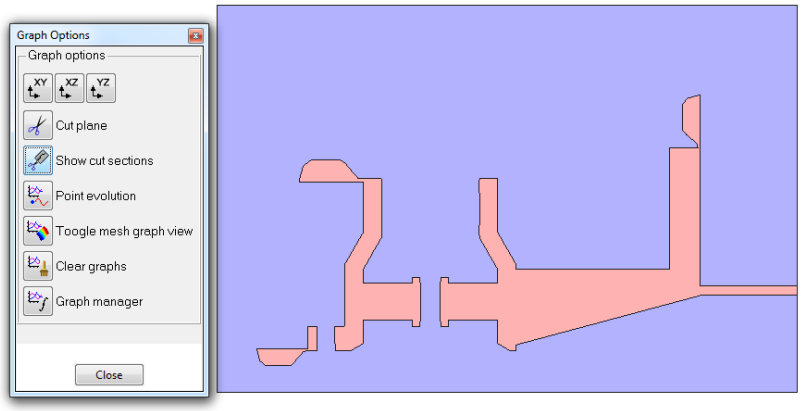


Figure G.42. Temperature in cut.

On the figure, we can see the temperature evolution of both the casting and the mould, in the **Cut xz plane**. By pressing the **Play** button on the animate window, we can see all the temperature evolution during solidification (select render normal to see a better resolution).

Now, we will add a virtual thermocouple in a point in the middle of the casting.

Go to:

View results → **Graphs** → **Point evolution** → **Temperatures** or select the button

And select a point in the middle of the casting :

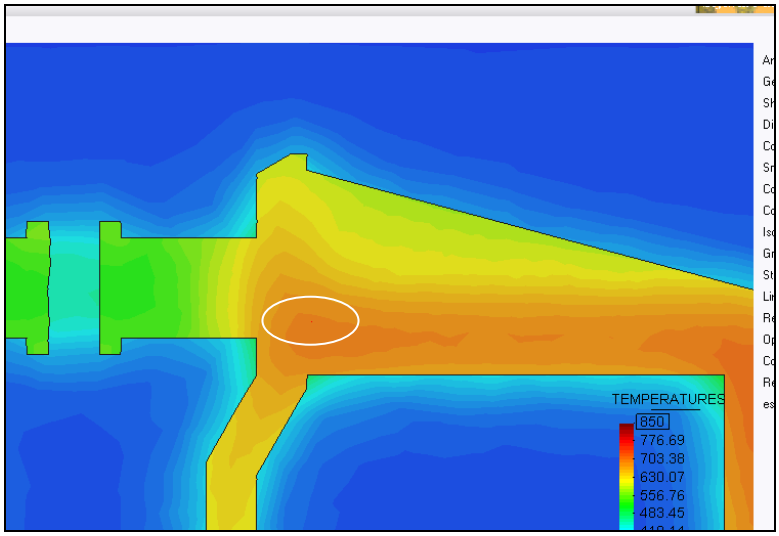


Figure G.43. Point selection.

The result will be:

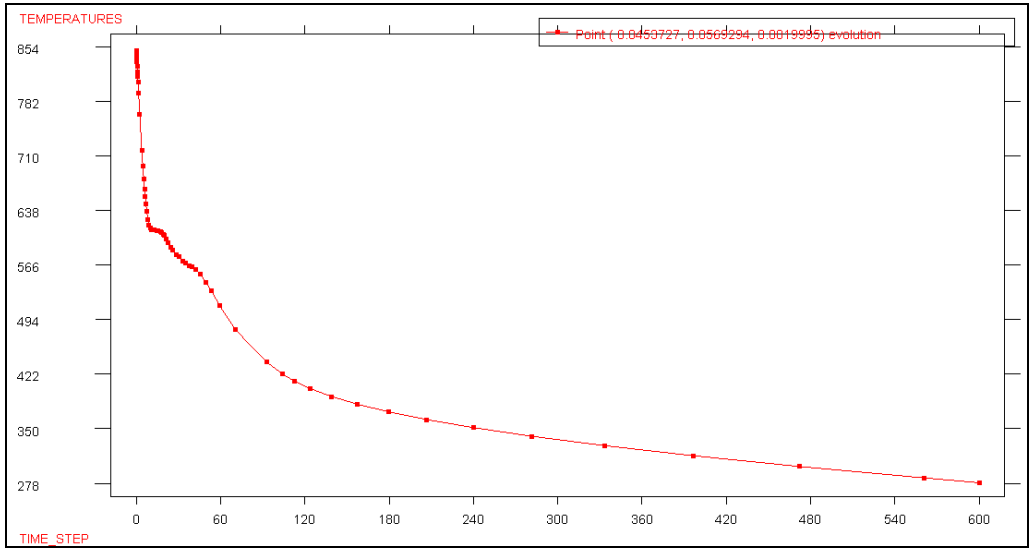


Figure G.44. Virtual thermocouple in the casting.

In the previous temperature images we cannot see the casting phase change, from liquid to solid. In order to see this and, for example, the effectivity of our riser system, the evolution of the solid phase, the last points in the casting to reach solid state, etc, we will do the following:

Go to:

Vulcan results → Solid fraction evolution or select the button

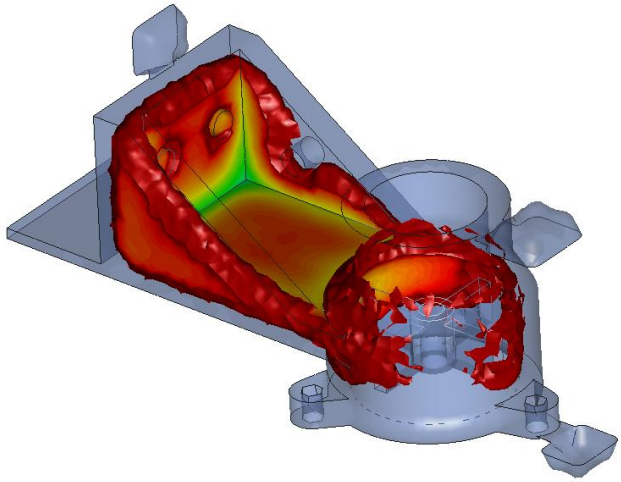


Figure G.45. Solid fraction evolution.

Here we have the liquid fraction of the casting is plotted in red, and the solid part of the casting disappears while is forming, so we only see the remaining liquid parts inside the casting as it solidifies, during the animation. The last points remaining in red should be the risers.

Finally, we will plot the **solidification modulus** of our casting. The modulus is the volume-to-surface ratio of the casting, and it can also be seen inside the casting by making cuts. Let's go back to ***Figure G.49*** and make again the temperature evolution, and then again temperature in cut. Once we have the results like in ***Figure G.42***, go to:

View results → Contour fill → Solidif Modulus

The result will be:

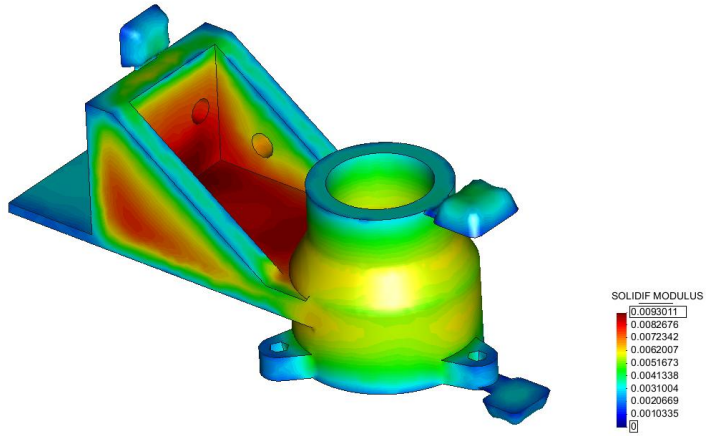


Figure G.46. Solidification modulus.

H. LPDC

LOW PRESSURE DIE CASTING SIMULATION USING VULCAN

Version 10.0

[This page is intentionally left blank]

INTRODUCTION

In this tutorial, we will follow an entire step-by-step procedure in order to run a **low pressure die casting simulation** ([1]) using Finite Element Analysis with Vulcan.

We will import an **IGES** geometry into Vulcan's graphical interface, mesh it, set the process parameters, and finally get simulation results (temperatures, velocities, turbulences, porosity, defects, etc.)

Basically, a **Finite Element Analysis** consists of three phases:

- Pre-process
- Calculation
- Post-Process

During the **pre-process** phase, we will completely define the problem geometry, set the types of analyses to perform, and establish the physical properties of the materials, temperatures, etc.

Once the problem is completely defined, we will launch the **calculation**; the total calculation time will depend on the complexity of the problem (this is, the number of **elements** in both the casting and the mould). During the calculation time, user's participation is not required. The calculation will let us know when is finished.

When calculation is finished, the user can start the **post-processing** phase; this is the results loading, view and analysis; this is the objective of the finite element simulation.

This tutorial consists of 9 steps, that will guide you trough all the low pressure die casting simulation process.

Before we start, let's describe Vulcan 10.0 interface. If you have previously worked with the new interface, you can skip Step 0 and continue to the next section.

NOTE ([1])*: The "**low pressure die casting**" is the process in which gas pressure is injected into the crucible, and the filling material goes up trough a refractory stalk, often used in the production of automobile wheels, axysymmetric parts, etc.*

Step 0

THE PROCESS BAR

Vulcan has a process bar ([2]), that will guide us trough all the preparations, settings and calculation of our casting process. This process bar is normally located on the left side of the screen:

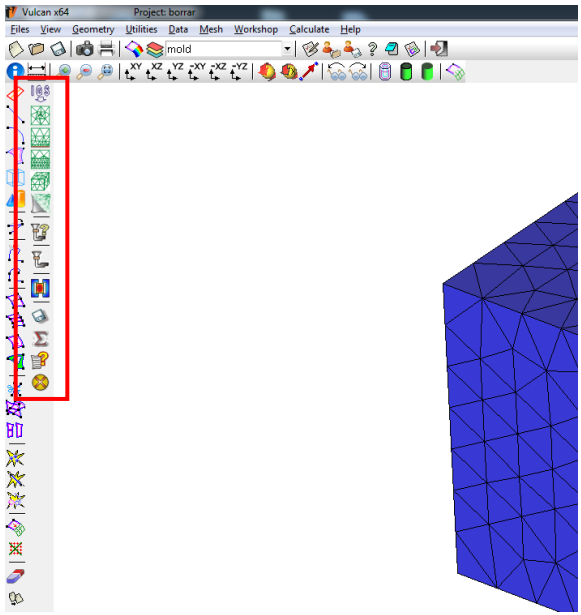


Figure H.1. The process bar.

NOTE ([2])*: If not present, you can activate the Process bar from the menu Utilities → Tools →Toolbars and also change its position to top, bottom, right, etc.*

Process bar icons

1. IGES read
2. Collapse entities
3. Mesh properties
4. Define the main process characteristic
5. Define foundry components
6. Define operations
7. Save the project
8. Calculate
   - 8.1. Process info
   - 8.2. Kill the process

These icons are placed in a logical order:

1 to 6	Pre-process
7	Save the project before calculate
8	Calculation

We can completely define and calculate the casting component under study using this bar, and in case necessary, interrupt the calculation by clicking icon 8.2.

Let’s begin by opening Vulcan and then with the step-by-step guided tutorial.

On the computer Desktop, double-click on the **Vulcan** icon:

the main screen will show up:

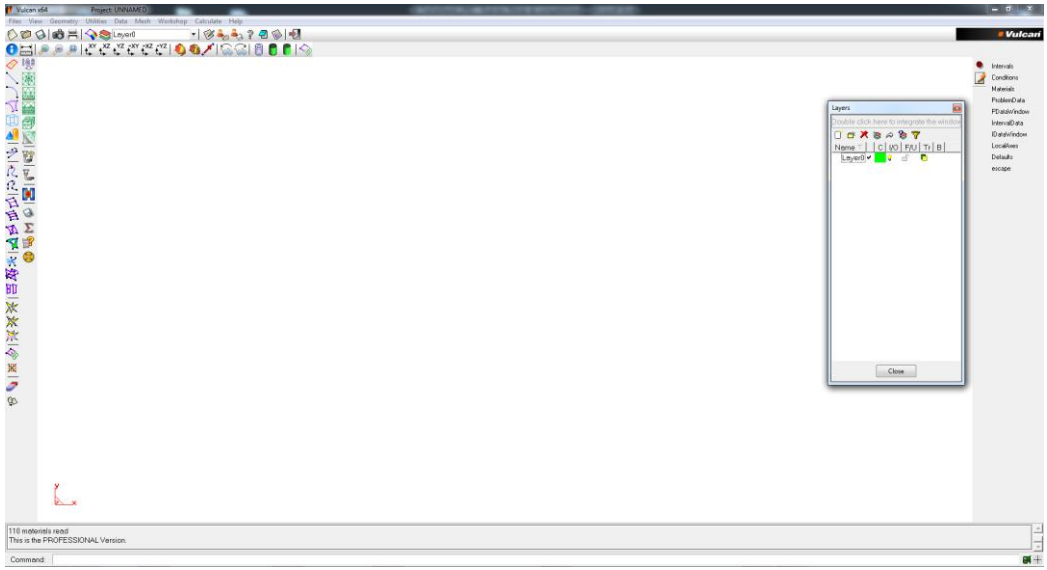


Figure H.2. Vulcan main screen.

Once Vulcan is opened, go to the main menu and click on:

Data → Problemtype → Vulcan

The following window will appear (***Figure H.3***):
(If the window doesn't appear, you can go directly to Step 1)

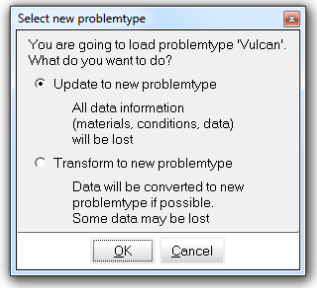


Figure H.3. Vulcan problemtype window.

Click OK and the process bar will open automatically; it will be used and described in detail in the present tutorial.

Now that we have the Vulcan pre-process interface opened and set, let's begin with the first step.

Step 1

READ AN IGES FILE

Before reading the IGES file, we are going to set Vulcan parameters in the optimal configuration for importing this type of files. In the main menu, click on **Utilities → Preferences…**
The following window will open:

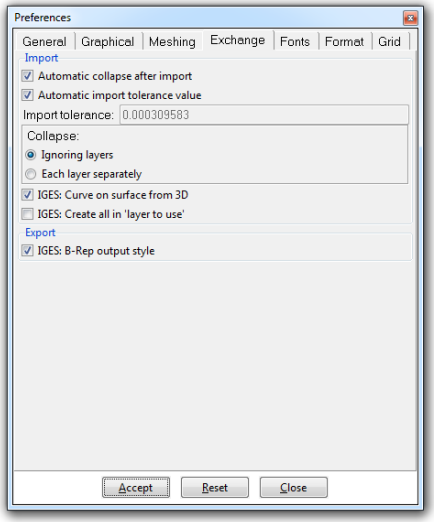


Figure H.4. Setting importing preferences.

The following checkboxes must be selected in ***Figure H.4***:

- ☑ Automatic collapse after import
- ☑ Automatic import tolerance value

Accept the changes and **Close** the window.

Now we are ready to import IGES files. Let's import a sample file.

Click the first icon on the process bar:

and read the following file:

"piezalpdc.igs" and click **Open**.

This file can be downloaded from the VULCAN web page http://www.quantech.es/support-vulcan.aspx

Select it and click **Open**.

the result should look like this:

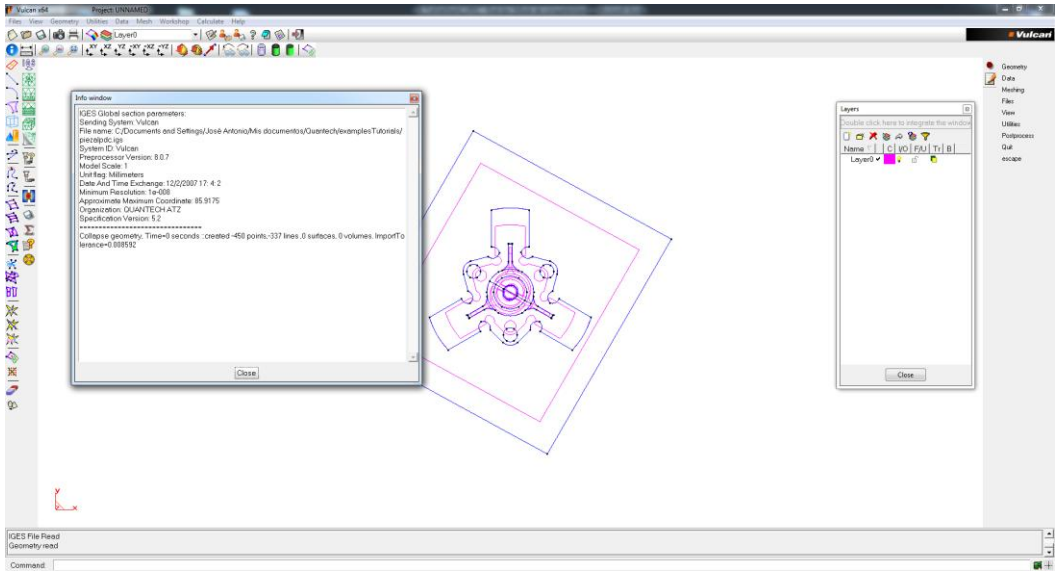


Figure H.5. IGES read.

Close the info window and have a look at the geometry of ***Figure H.5*** by right-clicking and doing **rotate-trackball** and **zoom in** and **out**.

Once done with looking at geometry, we will start working with different layers: one layer for the casting, one layer for the mould, etc. Click on the Layers icon:

The following window will appear (***Figure H.6***):

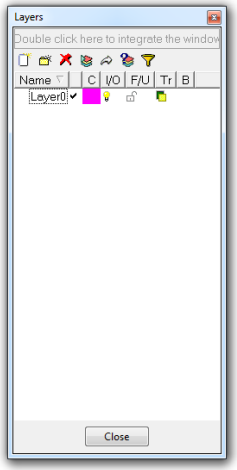


Figure H.6. Layers control window.

Perform the following operations:

- Click on **New** button, and create a new layer.

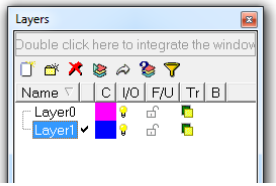


Figure H.7. Layer selection and layer to use.([3])

- **Select** the layer Layer0, enter the name “cast” in the textbox and click on **Rename** button.

NOTE ([3]) The **selected** layer is highlighted in black. After selecting a layer by simply clicking on it, we can delete it, rename it, change its color, turn it on, and off, send elements to it, etc. The **layer to use**, instead, is the layer with the checkmark: in the layer to use we can create geometry, delete it, etc. in the drawing. Summary: **selected** is to make changes using layers window, and **layer to use** is the layer used to create, delete, or modify geometry.

- **Select** the layer Layer1, enter the name “mould” in the textbox by clicking on **Rename** button.
- Be sure that **Layer to use** is cast; it says so beside the button, and there is a checkmark (✓) in the layer name.
- Select the layer mould, and click on **Send To** button, and then Surfaces.
- Go to the casting geometry and select the six surfaces that form the exterior of the mould.
- Turn **off** the layer mould.
- Select the layer cast, and then **Send To** and select all the cast surfaces; then press **Esc**
- **Close** the layers window.

The resulting geometry at this moment shall look like ***Figure H.8***.

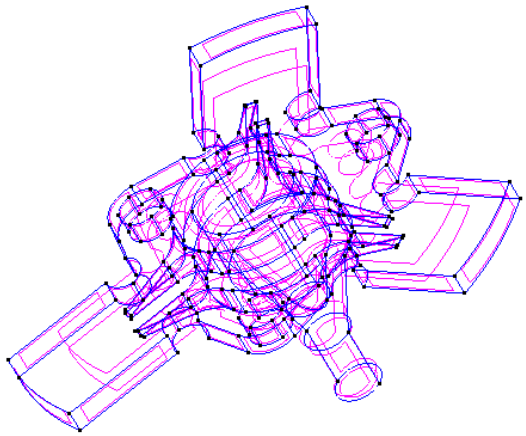


Figure H.8 Gravity casting geometry.

The geometrical entities in Vulcan have the following hierarchy order:

Points
Lines
Surfaces
Volumes

We can interpret the hierarchy by higher entities and lower entities. For example, a surface is higher than a line, a surface is lower than a volume, etc; we cannot delete a surface if there are related higher entities present (Volumes).

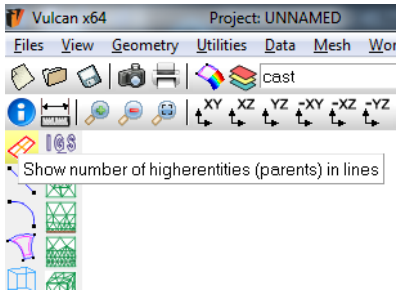
In ***Figure H.8*** we can see the points in black, lines in blue and surfaces in pink. There are always points that define line ends, lines that define surfaces, etc.

Now that we have a geometry to work with, let's begin with the next step, geometry treatment.

Step 2

GEOMETRY TREATMENT

This geometry apparently has no errors, so we won't collapse points or lines. Let's check the integrity of the geometry by selecting **Draw higher** option. Once the three-icon submenu is open, click on the **Draw Higher** icon:


Vulcan x64
Project: UNNAMED
Files View Geometry Utilities Data Mesh
cast
XY XZ YZ -XY -XZ -YZ
Show number of higherentities (parents) in lines

The result shall look like this:

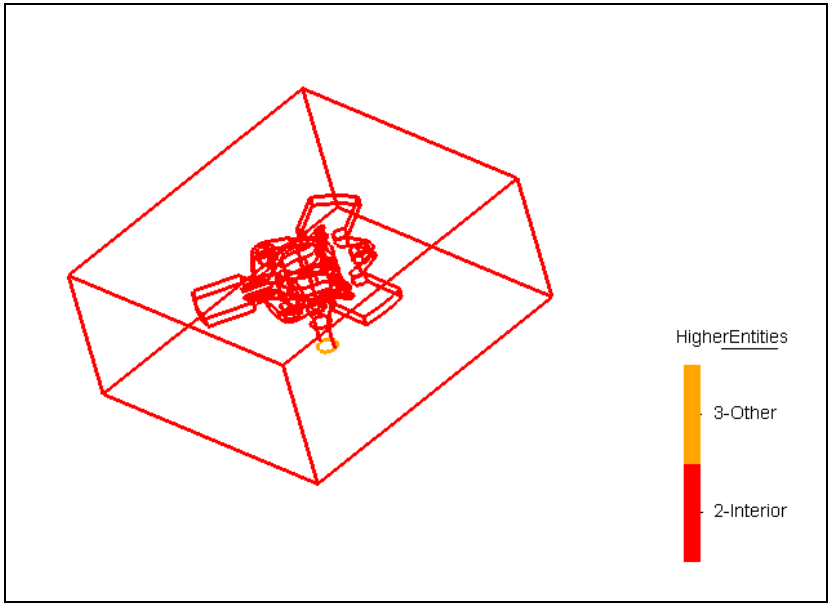


Figure H.9. Draw Higher screen.

In ***Figure H.9*** we can check the geometrical integrity of the model before we mesh it. To form a closed surface, all the lines need to be the end of two surfaces (“Interior” in the figure). If there would be open edges on the geometry, the lines would appear under a different colour and labelled “Boundary”, or “Isolated” lines.

The lines at the entrance of the material (“Other”) are connected to more than two surfaces because they are also connected to the mould surfaces, so they have a different colour; this is correct. The other lines are all “interior” (in red). This means that all the lines are connected to two surfaces (the lines are not forming open boundaries or are isolated lines). In other words, this geometry, consisting of points, lines and surfaces, is sound.

We can rotate, and zoom the casting while we are in **draw higher** mode.

Now let’s press **Esc** to go back to the geometry.

We have now a repaired geometry. Let’s move on to the next step.

Step 3

MESH GENERATION

Before start the meshing generation, we will set the Meshing parameters as shown in **¡Error! No se encuentra el origen de la referencia.**.Go to **Utilities** → **Preferences** and select the label **Meshing**. Set all the parameters as shown in the figure:

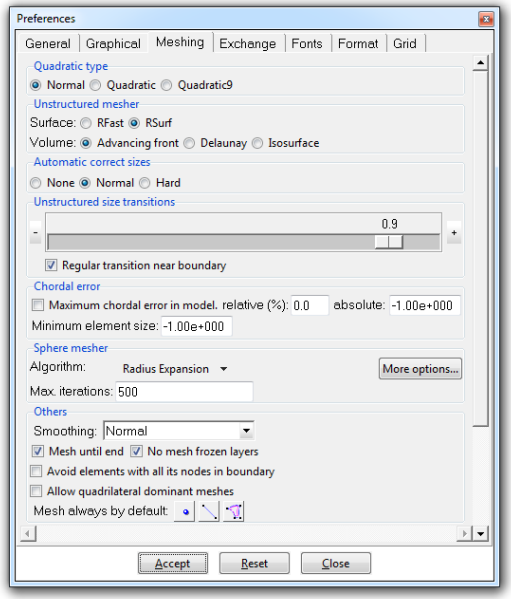


Figure H.5 Meshing preferences.

Click **Accept** and then **Close** the window.

It is recommendable to first generate a (triangular) surface mesh in order to generate a good (tetrahedral) 3D mesh, because this way we can check the mesh quality faster. The steps to generate a good tetrahedral mesh will be:

- Generating a surface mesh
- Check the quality of the mesh generated and make corrections, if necessary
- Return to geometry view and generate a geometrical volume out of the casting surfaces
- Generate a tetrahedral mesh of the casting, overwriting the previous mesh
- Check again for mesh quality
- Return to geometry and generate the mould volume, and then generate the final casting and mould mesh, overwriting the previous mesh; this is the only mesh needed; the previous ones were generated for verification

Save the project before meshing. Go to **File → Save as…**

Go to the Mesh menu as shown in ***Figure H.6***:

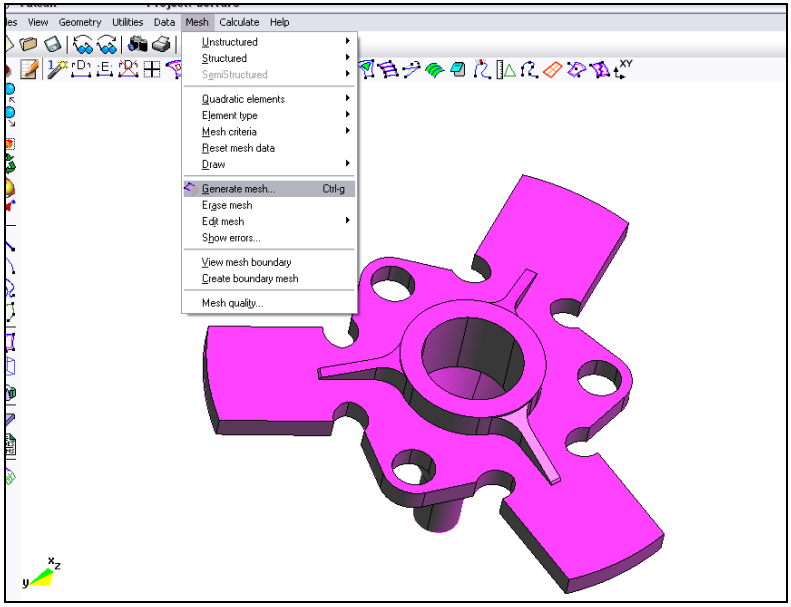


Figure H.6. Mesh generation screen.

The automatic assigned size of the mesh is **12.5**; change this value to **6.0** and click **OK**. The result will be like this: ([4])

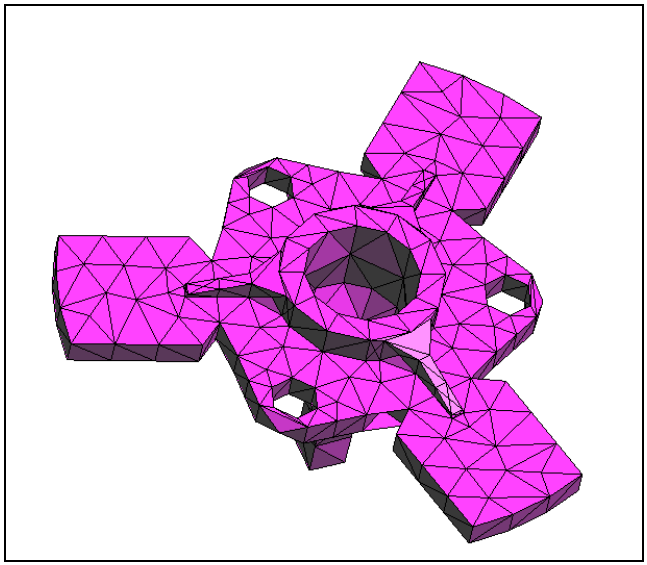
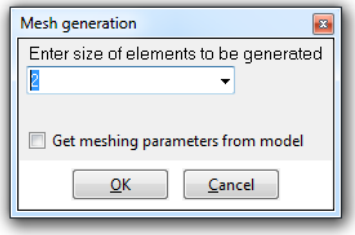


Figure H.7. Preliminary triangle mesh.

We can see in ***Figure H.7*** that elements are too big in order to be acceptable. Let's decrease the size of the elements to obtain a more uniform mesh.

NOTE ([4])*: We can change between geometry view and mesh view by clicking on the icon:* *and also change normal view to render view by right-clicking on the geometry and selecting* ***Render →Flat*** *or* ***Render →Normal***

Go to Mesh → Generate Mesh…
And set a mesh size of **2**:



The result should be like in ***Figure H.8***:

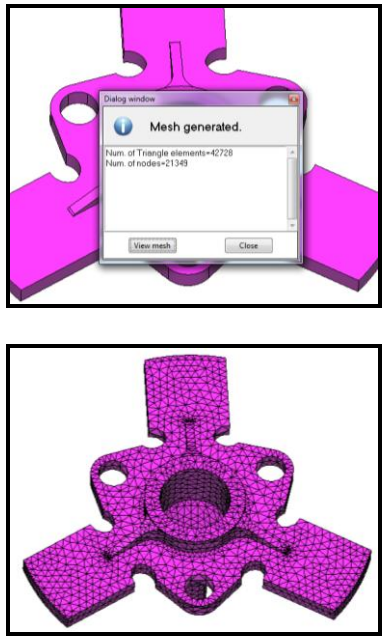


Figure H.8. Mesh generation.

Compare the results of ***Figure H.8*** and ***Figure H.7***, and look at the notorious difference at the element sizes. ***Figure H.8*** represents much better the real casting geometry.

Let's now return to geometry by clicking and generate the volume and tetrahedral elements. Go to Geometry → Create → Volume → By contour or click the **Create volume** icon .

The following will appear in the command line:
"Enter surfaces to define volume (ESC to leave)"

Select all the surfaces of the casting with a selection window ([5]):
"Added 114 new surfaces to the selection. Enter more surfaces. (ESC to leave)"
press **Esc**

"Created 1 new volume. Enter more volumes"
And now press **Esc** again:
"Leaving volume generation"

Now regenerate a mesh with size of 1 in order to obtain a mesh of the volume. The result will look again as in ***Figure H.8***.

Generating the mould mesh

Return to geometry view. Click on Layers icon, and turn on the layer **mould**. With the layer mould selected, click on the **Layer to use:** icon.

Now, we will generate a volume for the mould.

Go to **Create volume**, and select all the surfaces with a selection window:

NOTE ([5]): We can use the Layers icon to facilitate the selection of the casting surfaces. Leave the **cast** layer on and the **mould** layer off for this selection.

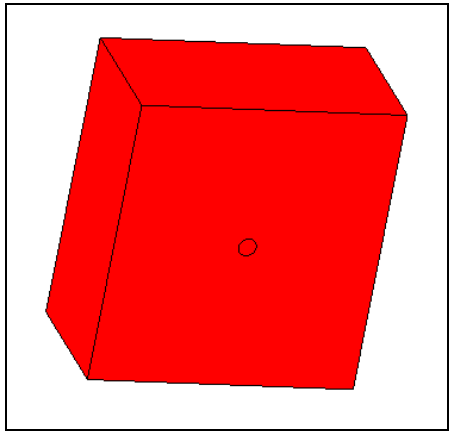


Figure H.9. Mould volume generation.

Now zoom on the small circle (material entrance) and click on the small surface in order to deselect it. The result should look like this:

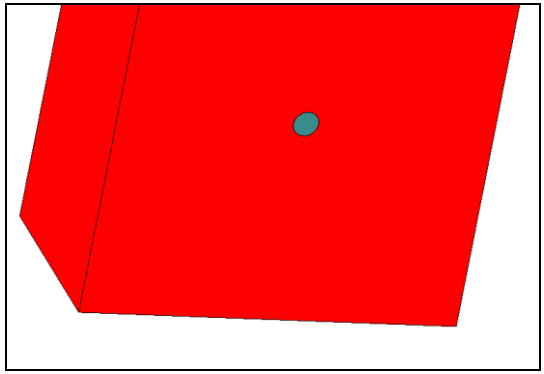


Figure H.10. Mould volume generation.

Now press **Esc** twice and the mould volume will be generated.

Assigning mesh size to the mould

In the case of the mould geometry, we will try to use tetrahedral elements as big as possible, in order to reduce the total number of finite elements of the problem. This will reduce the calculation time; so, instead of assigning an automatic size of **2** to all the geometry (cast + mould) we are going to do the following:

Turn off the layer **cast**.

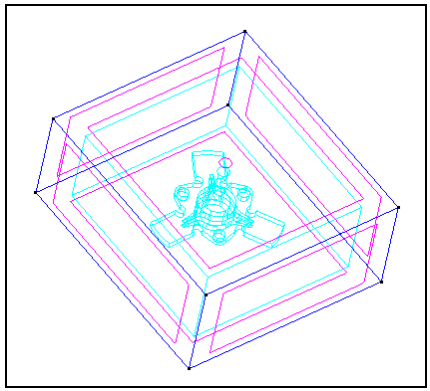


Figure H.11. Mould layer.

And now, we will see in detail how to assign properties to the mesh.

Using the **assign mesh sizes** icon, we will assign properties to the geometrical entities of our geometry:

We have to assign mesh sizes in geometrical hierarchy order: **Points, Lines, Surfaces and Volumes.**
Let's assign a mesh size of **10** to all the entities of the mould:

a) Click on the first icon .

The following window will open:

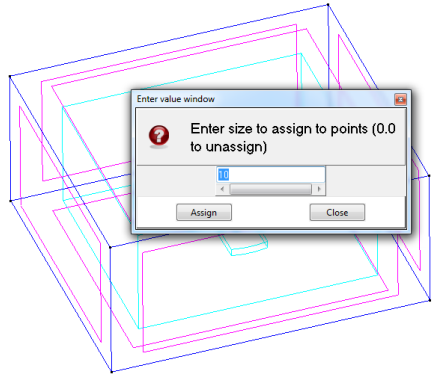


Figure H.12. Mesh generation

b) Assign a size of **7** in the window and click the **Assign** button.
c) Select all the points of the mould, with a selection window:

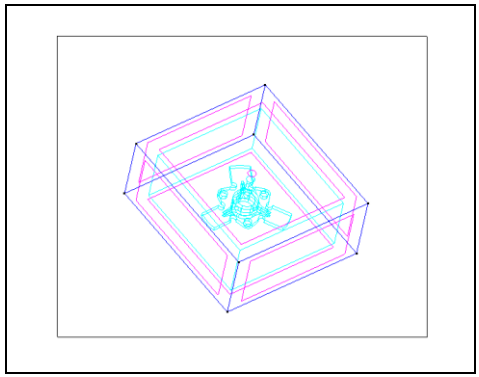


Figure H.13. Selection window

d) In the command line, the following message will be displayed:
"Added 8 new points to the selection. Enter more points. (ESC to leave)"

These are all the points added to the selection; the points are added, and as it says in the command line, we have to press **Esc** to accept the selection and leave.
The following message will appear in the command line:

"Assigned size=10 to 8 new entities"

e) click **Close** to close the enter value window.

Now we repeat **points a) to e)** for lines, surfaces and volumes, with the same size of 7.

Once the sizes are all assigned, we generate the mesh again (same general mesh size: 2). The result shall look like this (***Figure H.19***):

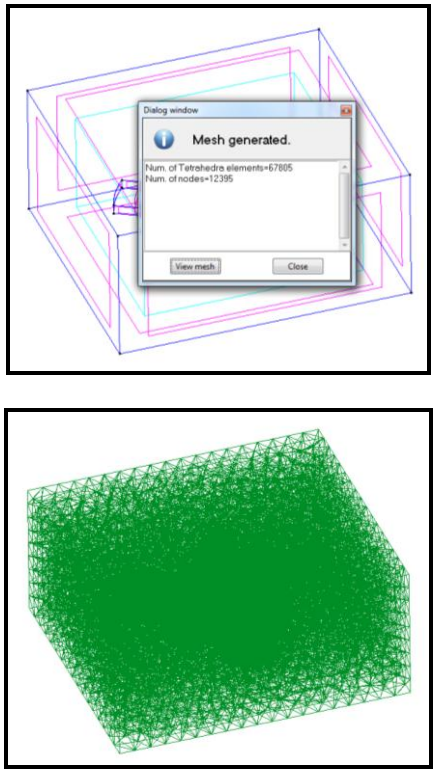


Figure H.19. Generated mesh

Now that the mesh is generated, we will see in detail how to set the process parameters for this particular type of problem, low pressure die casting.

We will continue the step-by-step process with the **Process bar**.

Step 4

FOUNDRY PROCESS DEFINTION

Until now we were dealing with geometrical parameters, definitions, etc.

From now on, we will deal with real foundry parameters, characteristics, processes, etc. It is very important to set the parameters properly. The closer the parameters to reality, the more accurate results. So read on and let's do casting simulation.

The first step we take once we have the casting and mould geometry defined and meshed, is let Vulcan know what type of process we are going to calculate. Click on **define the main process characteristic**, the 4th icon:

A window with 3 labels will appear. In the **Process type** label, select **Low pressure,** as shown in ***Figure H.14***.

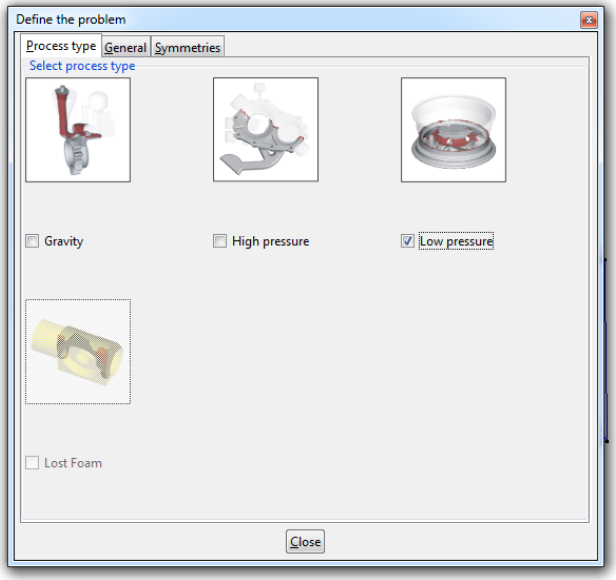


Figure H.14. The Process type window

Once this step is taken, Vulcan adjusts all the internal parameters and the remaining icons to this particular type of process.

In the **General** label, we have a set options to define: units, gravity direction (with respect to the coordinate system compass), environment temperature, etc.

Set the gravity direction to –Z, and the Environment temperature to 25 Degrees Celsius, as shown in ***Figure H.15***:

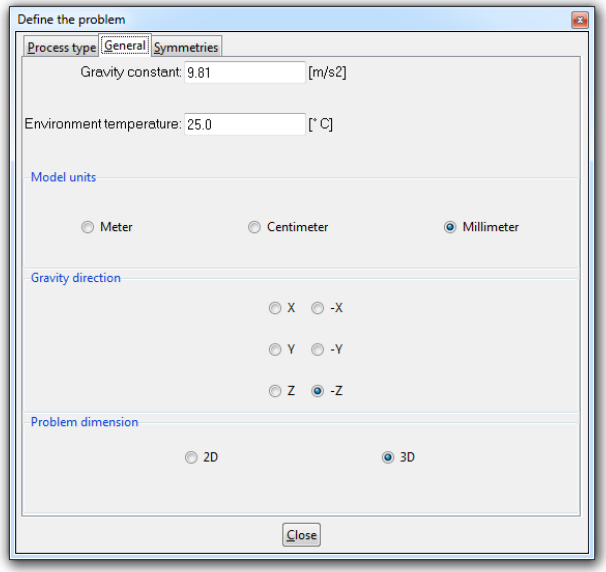


Figure H.15. The Process type – General window

In the **Symmetries** label (***Figure H.16***) we have to define the symmetry planes created; if there are no planes of symmetry, simply leave **No planes** selected:

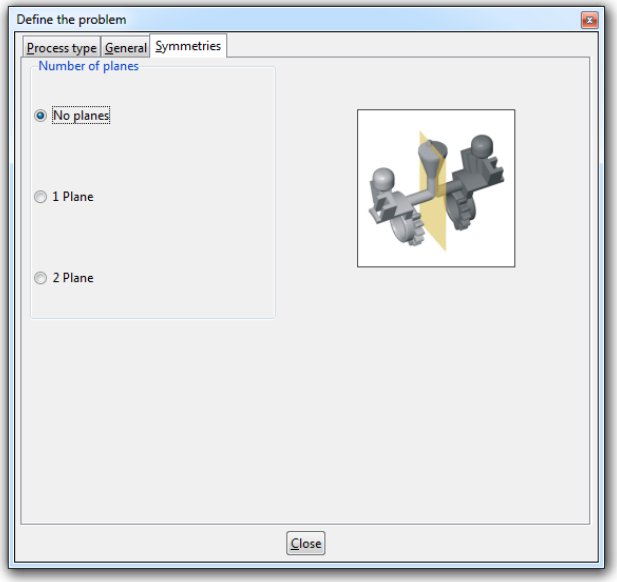


Figure H.16. The Process type – Symmetries window

Now the gravity casting process is defined; click **Close** ([6]) to continue to the next step.

NOTE ([6]): *All the changes made will be saved upon clicking on the **Close** button.*

Step 5

FOUNDRY COMPONENTS DEFINTION

Click on the **Components definition** icon:

The following window will open (***Figure H.17***):

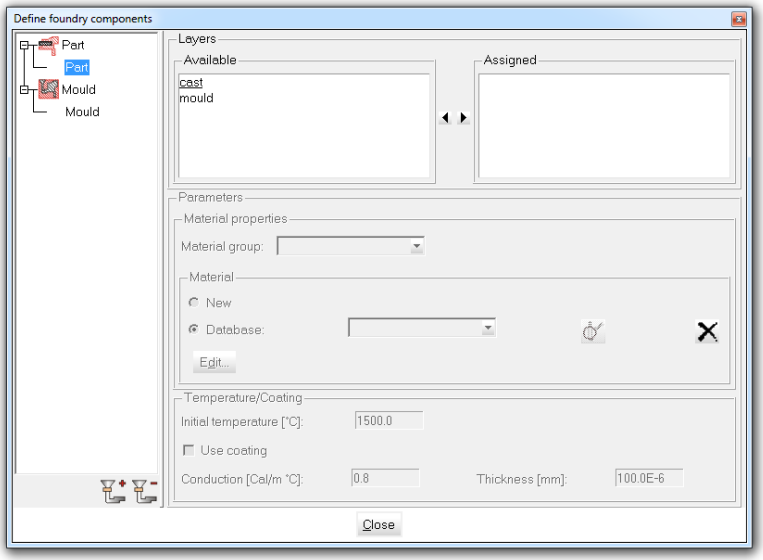


Figure H.17. Foundry components definition.

First we have to assign each layer to the corresponding foundry component. On the example shown, the Part component is highlighted on the tree. In the **Available** window, select **cast** and click on the Icon. The **cast** layer will move to the **Assigned** window.

Now, select the Material group and specific material from the database, and set a temperature of 640 Degrees Celsius, as shown in ***Figure H.18***:

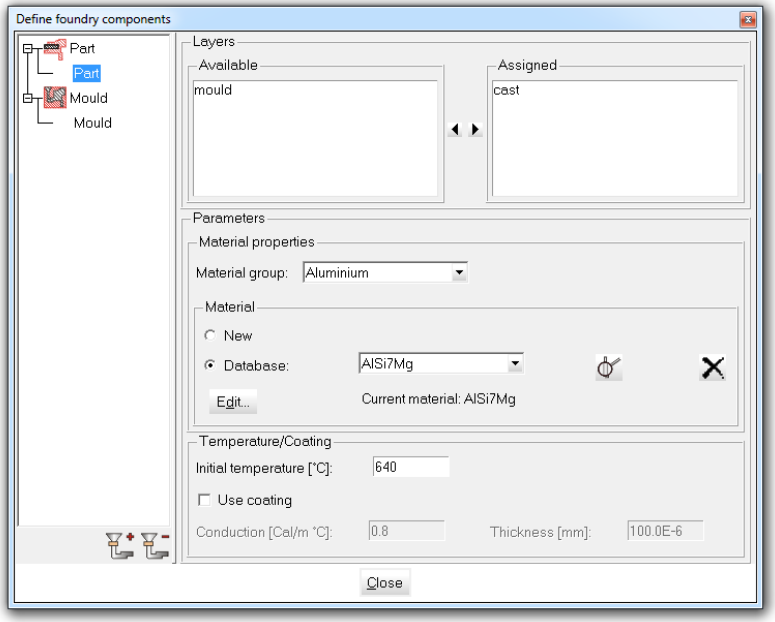


Figure H.18. Foundry components definition.

Now perform the same procedure for the mould, first selecting the **Mould** component on the tree, and then **mould** from **Available** window. Again, it should move to **Assigned** window.

Assign in the **Mould materials** group, the **steel** material at initial temperature of 20 Degrees Celsius:

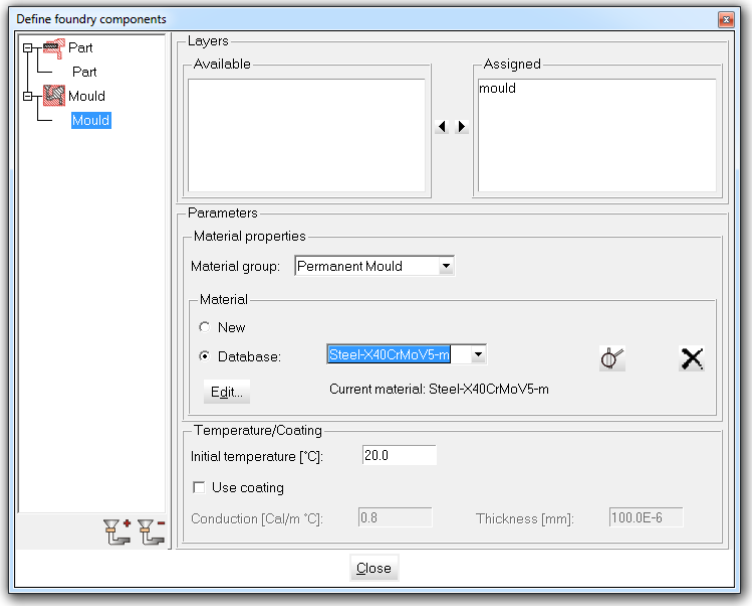


Figure H.19. Foundry components definition.

Click **Close** to save the changes.

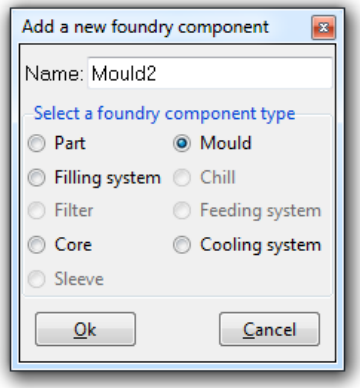
***NOTE**: Let's explain some of the icons that appear in the **Components definition** window. You can skip this note to continue with the example, and go to the Step 6.*

Click again on the icon:

In case of having multiple layers, you can assign and unassign layers to foundry components by using the *icons (for example, in case of having the mould divided in upper part and lower part on different layers, or in case of having to make changes).*

*Now click on the **Add new foundry component** icon:*

The following window will appear:



This window has to be used in case of need to insert more components to the tree, such as more parts of the mould, or cores. On the example shown, there are only two layers, the whole mould and the part, so there is no need to create a new foundry component.

*In **Figure H.19** we can also see the **Use coating** checkbox. If we use this option, we can change the conductivity and the thickness for the specific coating we are using.*

*Now click on the **Edit…** button; this window will open:*

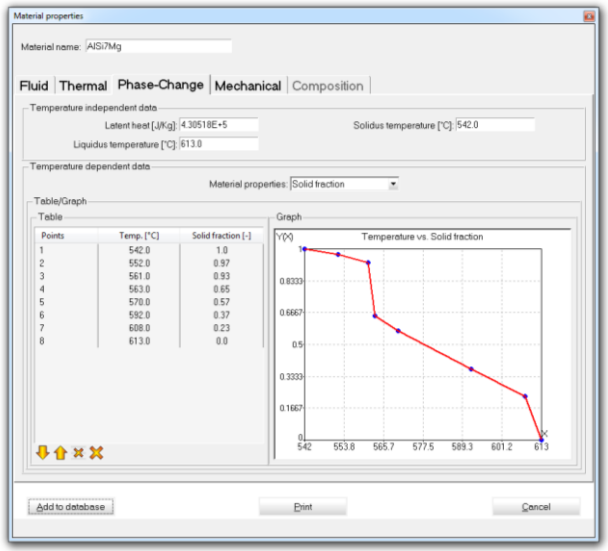


Figure H.20. Editing material properties

*Here we can see the material properties in both table and graphical views. We can change these data, add new materials, etc. See the Tutorials for more details. Now click **Cancel** on this window.*

*And click **Cancel** to continue with the example.*

Step 6

OPERATIONS DEFINITION

Now we are going to define the specific operations to be simulated. Let's click on the **Operations** icon:

The following window will appear (***Figure H.21***):

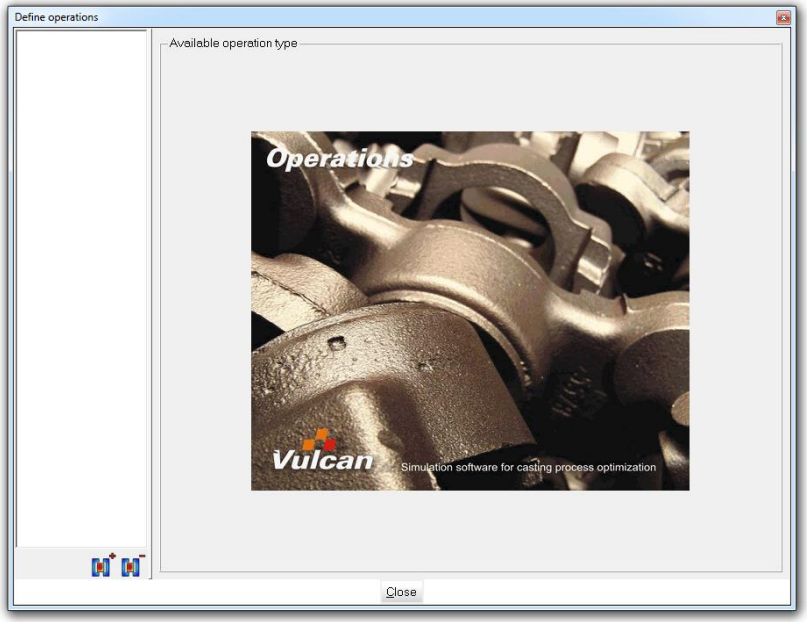


Figure H.21. Operations definition.

From the point of view of casting simulation, we can divide the casting process into three parts: **filling**, **solidification**, and **cooling**.

The filling begins when the material starts to enter into the mould, and finishes when the molten metal fills the entire mould.

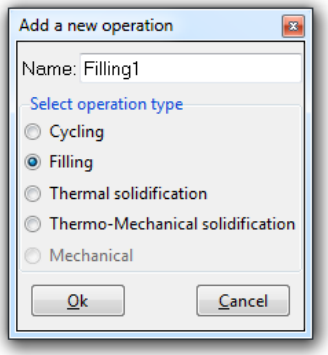
At this moment, solidification starts, as metal begins to cool down and solidify; solidification finishes when the last part of the casting turns into solid phase.

At this moment, cooling starts, and goes all the way down until the casting and mould reaches room temperature.

In our gravity casting example, we are going to run a simulation the **filling** and **solidification** parts of the process.

Click on the add operations icon:

The following window will appear:



Here we have to select all the operations we want to simulate one by one:
Leave Filling and click OK, and then enter on the same previous icon, and select **Thermal solidification** operation.

The result should look like this:

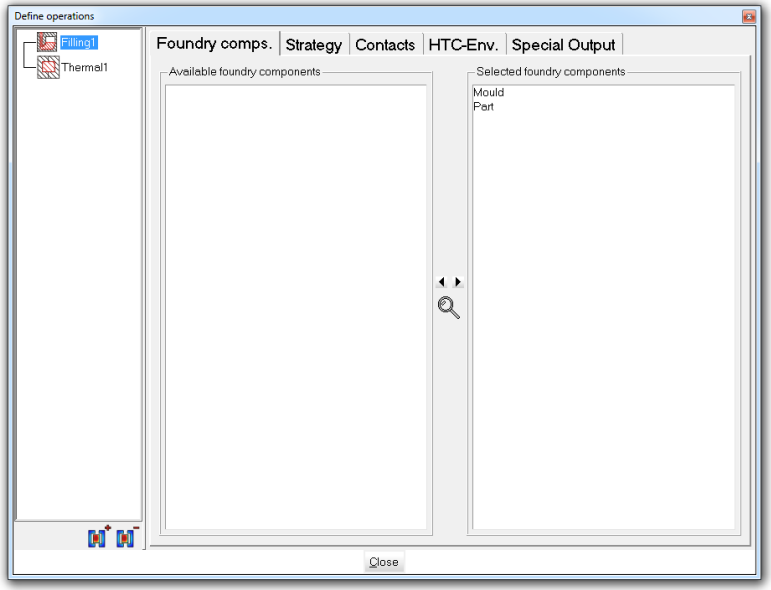


Figure H.22. Operations definition.

In ***Figure H.22*** we can see that all the foundry components are assigned by default to the filling and solidification operations. In (***Figure H.17***) we created those foundry components. Here we have the option of changing the foundry components, selecting different components for different operations, etc.

To continue with the example, we leave the default selection, and then, we are going to view and set all the parameters for both operations. We have to go trough all the labels and change the default parameters for those of interest. In our example, we will see all the options and a short explanation of the principal parameters.

In strategy label (***Figure H.29***), we set the parameters for the low pressure die casting process, and we decide whether to use tilt pouring or not. In case of using tilt pouring, we have to select it and enter all the options. In the figure, we can see that we have to set some parameters of the filling process; we have to enter the distance between the free surface of the crucible and the end of the filling stalk, and also the Time vs. Pressure curve of the filling machine.

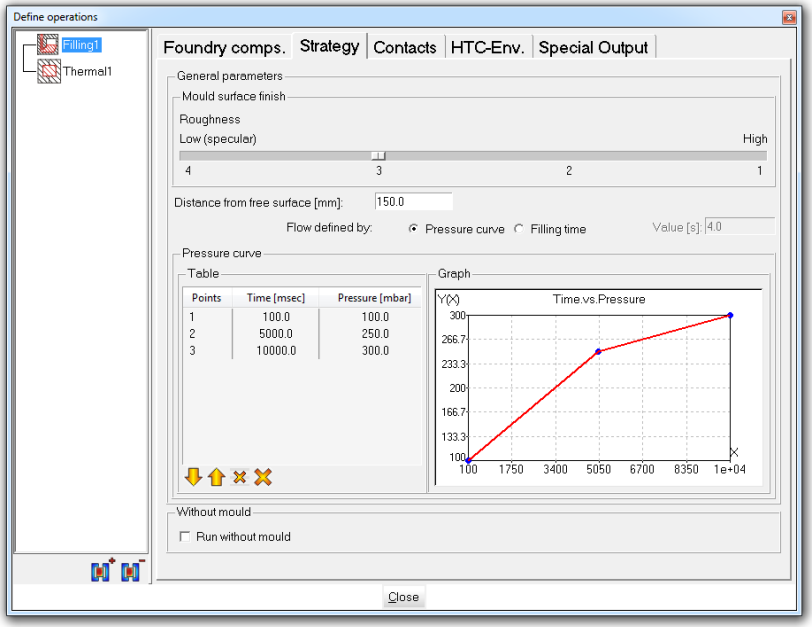


Figure H.29. Strategy label

Select the option “Filling time” in ***Figure H.29*** enter a value of 0.3 sec, and continue to the next label.

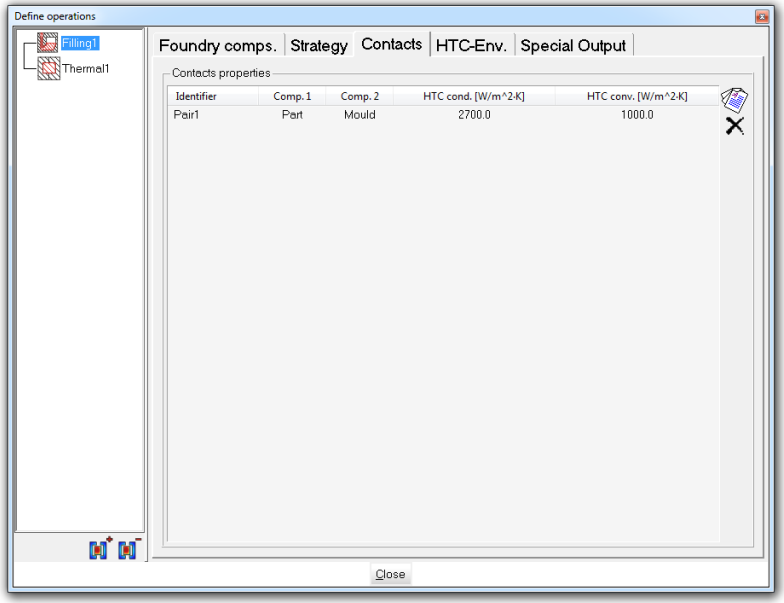


Figure H.23. Contacts label

In the Contacts label (***Figure H.23***), we set the values of the Heat Transfer Coefficients (HTC) for the contact between the casting and the mould. This contact is characterized by two different values: HTC conduction and HTC convection. Keep values to:

HTC conduction: 2000
HTC convection: 1000

And continue to the next label.

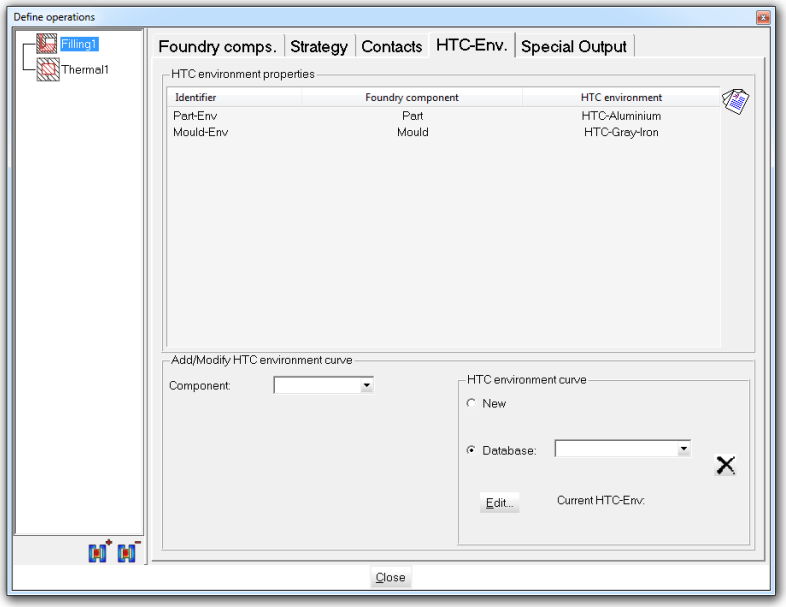


Figure H.24.HTC-Environment label

Let's have a look on ***Figure H.24***. Here we set the value of heat transfer coefficient with environment (HTC-Env) in this case, environment is the air at the temperature set on the problem definition ([7]).

To continue with the example, leave all the default options and continue to the next label.

NOTE ([7])*:* We can modify or create a new value for this parameter by entering on the Edit... button and setting a new value. The parameter could change, for example, if the air is not still (there is air circulation with fans)

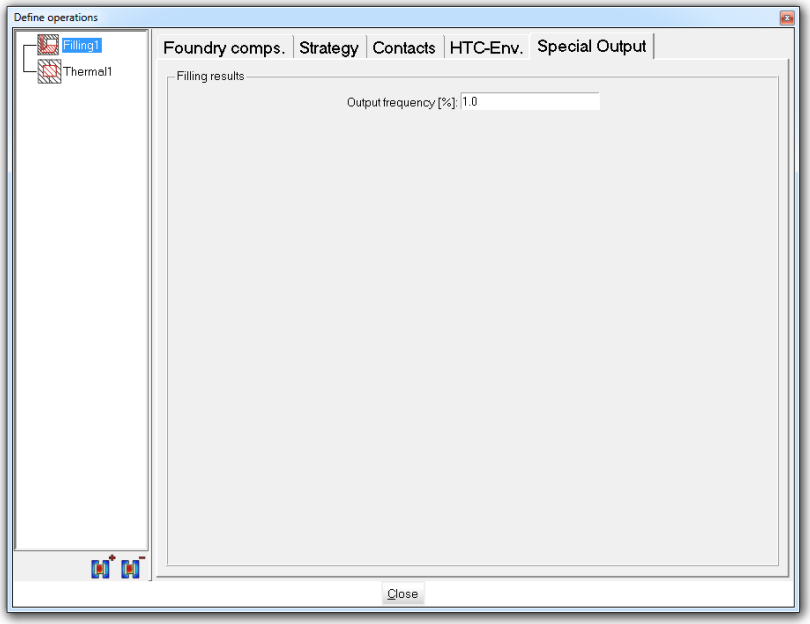


Figure H.25. Special Output label

In ***Figure H.25***, we define the number of results to be written during the analysis. Leave the default option.

Now, change to **Thermal1** operation on the operations tree, as shown in ***Figure H.26***:

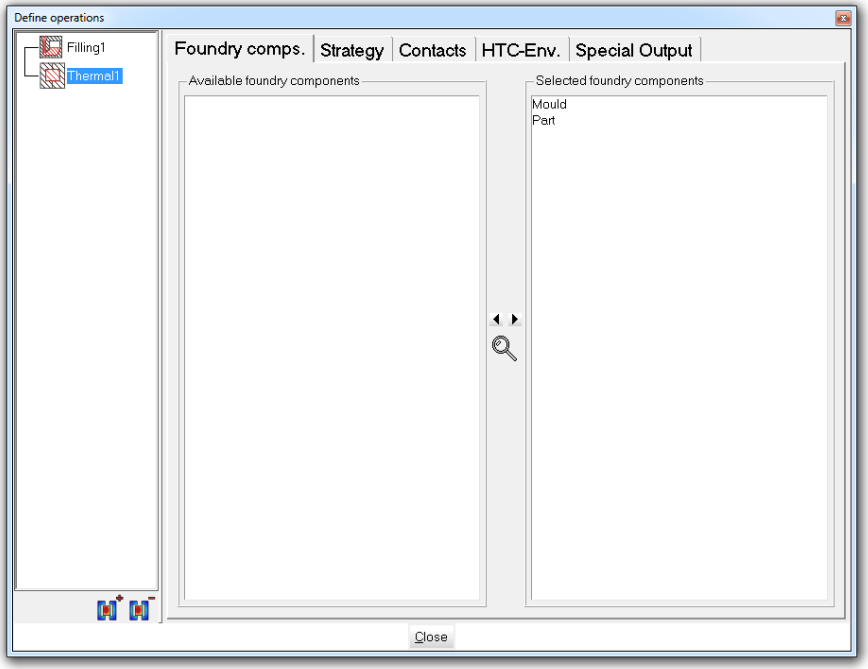


Figure H.26 ***Thermal1*** *settings*

In this screen, foundry components are already selected. Go to the next screen:

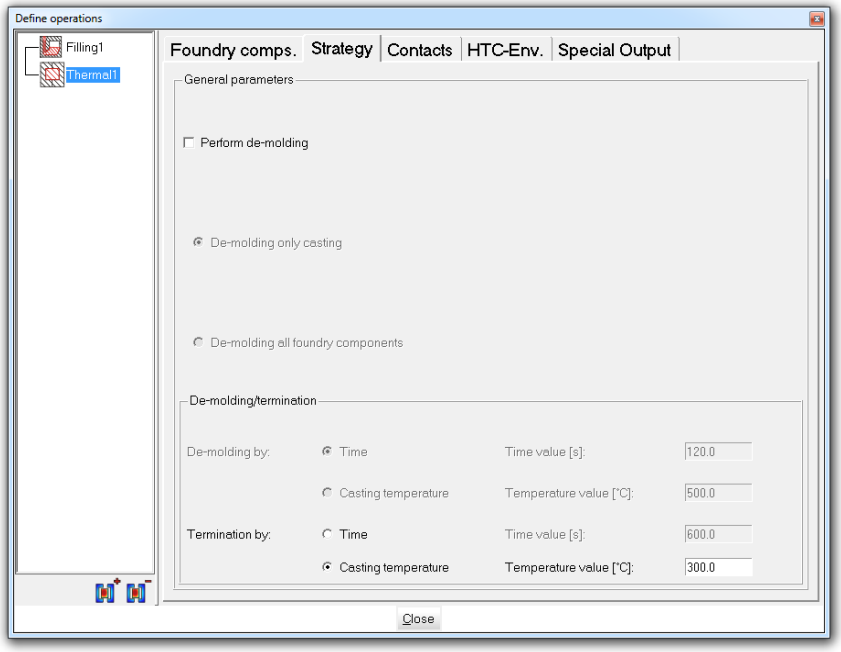


Figure H.27. Thermal Strategy label

In this screen, we define the **De-moulding** and **Termination** characteristics of our process. In the current problem, we leave de-moulding deselected, and we select termination by Casting temperature, 300 ºC. Click on the next label.

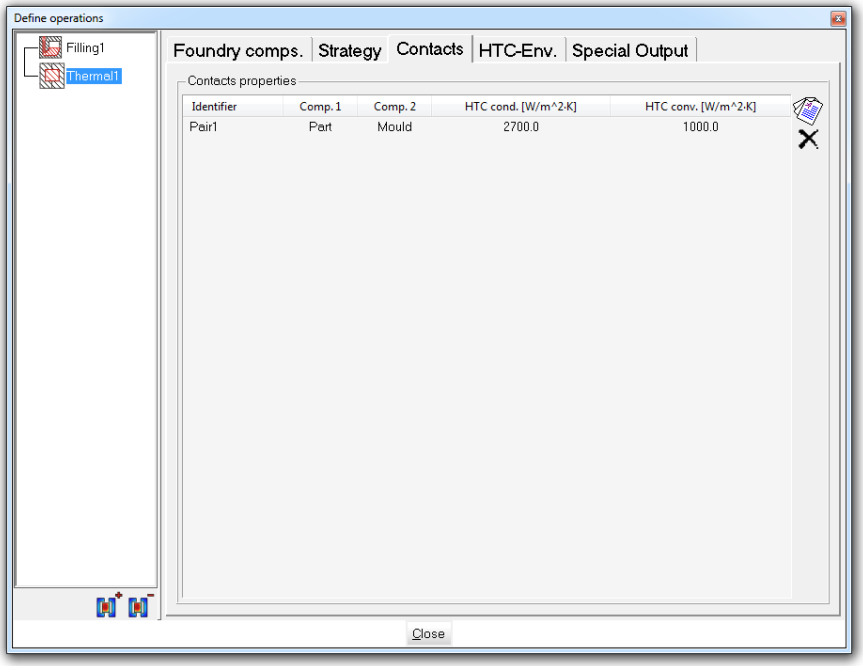


Figure H.28.Thermal Contacts label

Here again we set the HTC conduction to 2000, and HTC convection to 1000, and continue to the next label.

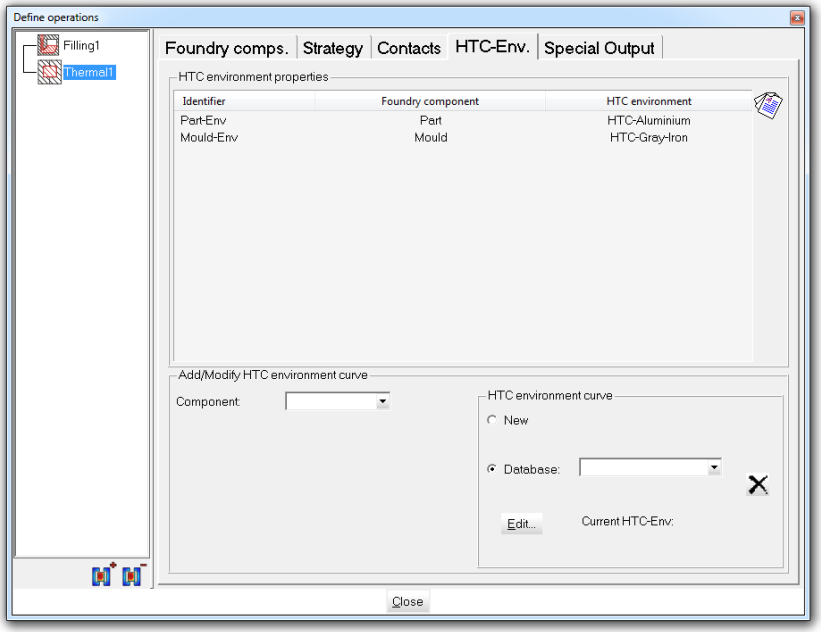


Figure H.29.HTC-Environment label

We have pre-selected the same options as for the filling analysis. Continue to the next label.

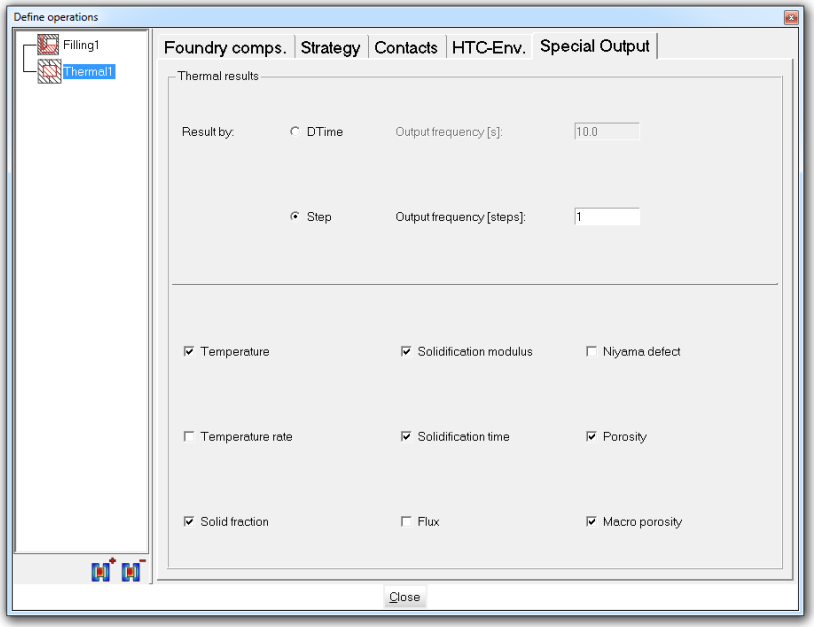


Figure H.30.Thermal Special Output label

In this label, we define the thermal results that we are going to visualize on the post-process. In the example, leave Temperature, Solid fraction, Solidification modulus, and Solidification time.

Now that we have defined all the problem parameters, click on the **Close** button. The parameters will be saved.

Step 7

SAVE PROJECT

Before start the calculation, don't forget to save the project:

Click on **Save** icon.

NOTE: the filename must not contain spaces or symbols.

Step 8

CALCULATION

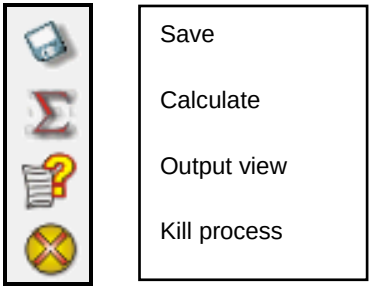


Figure H.31.Calculate, Output view and Kill Process icons.

Now, following the column of icons shown in ***Figure H.31***, we first saved the project and then **Calculate** it. The calculation time will depend upon the complexity of the problem.
The calculation window will look like this:

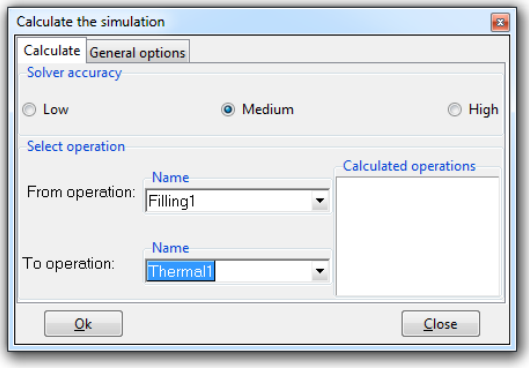


Figure H.39.Calculation window

Select **From operation** filling **To operation** Thermal1 and click OK. The calculation will start:

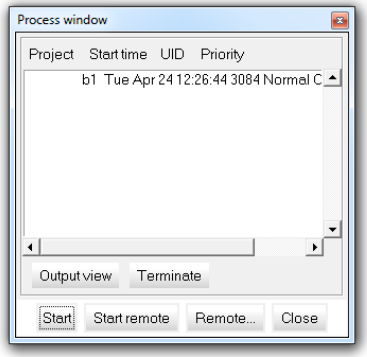


Figure H.320.Calculation window

We can view the **Output** view by clicking the icon and see the calculation status, and also interrupt and cancel the calculation by **Terminate** the process, make changes and **Calculate** again. Close the window.

OUTPUT VIEW

Once the window is closed, we can still access to the output view by clicking on the icon:

And we will see the calculation process:

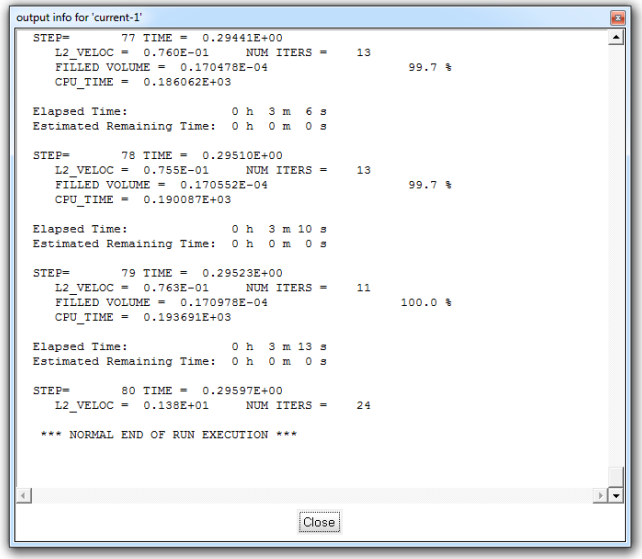

output info for 'current-1'
STEP= 77 TIME = 0.29441E+00
L2_VELOC = 0.760E-01 NUM ITERS = 13
FILLED VOLUME = 0.170478E-04 99.7 %
CPU_TIME = 0.186062E+03
Elapsed Time: 0 h 3 m 6 s
Estimated Remaining Time: 0 h 0 m 0 s
STEP= 78 TIME = 0.29510E+00
L2_VELOC = 0.755E-01 NUM ITERS = 13
FILLED VOLUME = 0.170552E-04 99.7 %
CPU_TIME = 0.190087E+03
Elapsed Time: 0 h 3 m 10 s
Estimated Remaining Time: 0 h 0 m 0 s
STEP= 79 TIME = 0.29523E+00
L2_VELOC = 0.763E-01 NUM ITERS = 11
FILLED VOLUME = 0.170978E-04 100.0 %
CPU_TIME = 0.193691E+03
Elapsed Time: 0 h 3 m 13 s
Estimated Remaining Time: 0 h 0 m 0 s
STEP= 80 TIME = 0.29597E+00
L2_VELOC = 0.138E+01 NUM ITERS = 24
*** NORMAL END OF RUN EXECUTION ***
Close

Figure H.33.Output view.

TERMINATE PROCESS

If we want to change parameters and calculate again, we would click on the last icon:

This icon will terminate the process. In the example, we will leave the computer calculating and wait for the end of calculation. We will not click on the icon.

Exiting Vulcan while is still calculating

When the pre-process is over and we are calculating a process, we can exit Vulcan application but without interrupting the calculation:

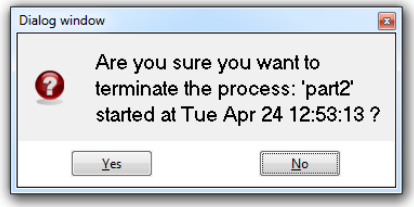


Figure H.34. Exiting Vulcan while is still calculating.

Click **No** and Vulcan will exit. We can open Vulcan later, during or after the calculation. The computer, of course, must remain on.

Step 9

POST-PROCESS

Post-process means the visualization of the simulation results; this is perhaps the most important step of all the casting simulation steps; here is where we read and analyze the simulation results.

Once the calculation is finished, we can visualize the results of this simulation. There are many different ways to do it, like colour maps, vectors, iso-surface curves, virtual thermocouples, etc.

Figure H.35 shows the window that appears once calculation step is finished; we can directly access to post-process by clicking on **Postprocess** button.

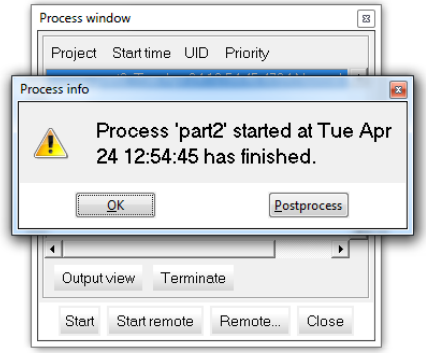


Figure H.35.Process info.

Another way to switch between pre- and post-process is by clicking the **toggle** ([8]) icon .

NOTE ([8]): Every time we toggle, when we return to post-process, we have to load the results.

The first step in results visualization is to load results of any **foundry operation** (Filling, thermal solidification, thermo-mechanical solidification). Let's load the filling operation and analyze the results.

Go to **Vulcan results** → **Load results**…

And select the **Filling1** operation:

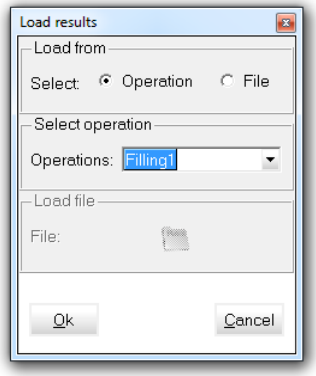


Figure H.36. Results loading

Click **Ok** and the filling results will be loaded.

Now that we have all the filling results loaded, let's begin, for example, to see the evolution of the casting material in the mould; where there could be air entrapment; or the advancing fronts of material could encounter; the last spots of the mould to be filled, etc.

To see this, we will use a tool called filling mater. In this animation, we will see the material free surface in red, the filled areas in grey, and the mould will be transparent.

Go to main menu:

Vulcan results → Filling mater

Once the Animate window appears, press the **Play** button:

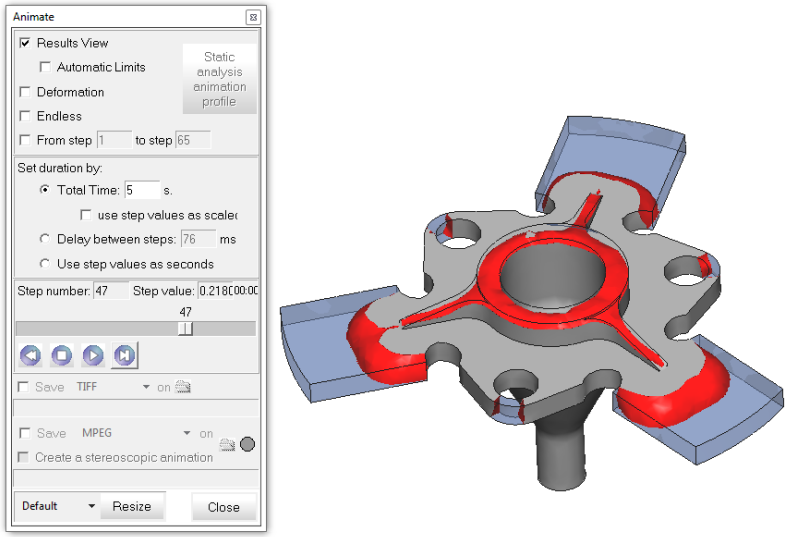


Figure H.37. Filling mater animation.

In ***Figure H.37*** we can see the **Animate** window and the control buttons. We can keep this window open while we rotate, zoom in and zoom out the casting. Let's explain the basic Animate controls.

Basic Animate window controls

Delay: We can set this time in order to “slow down” the animation; delay time is not the real time, but a time we can increase in order to ease the visualization.

Step: This box shows the real time step. It shows the time in seconds since the filling process had begun.

Then we have the animation controls:

these controls are the standard Play/Pause/Stop/Fast Forward/Rewind controls. We can control the evolution of the animation with them.

Once the filling mater analysis is done, go to:

View results → No results

Let's say now that we want to see other phenomena, for example, turbulences formation, highest filling velocities inside the casting, velocities at which the advancing fronts encounter, etc. We will use a different representation to see these results.

Go to:

Vulcan results → Filling vectors

And again, the animate window will show up. The animation will show the velocity vectors inside the casting:

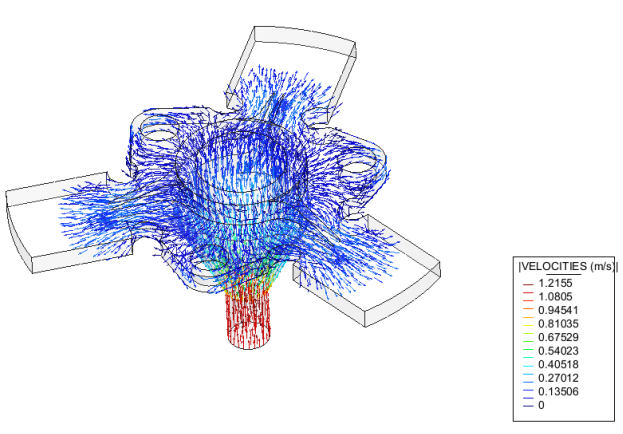


Figure H.38. Filling vectors animation

The colour scale represents the module of velocity. In ***Figure H.38*** we can see the highest velocities in red, and the lowest velocities in blue.

Once finished with filling vectors, go to:

View results → No results

Now we want to see parameters like temperature of front encounters, early cooling during filling, and possible solidification during filling.

To see this kind of results, we will go to:

Vulcan results → Temperature evolution

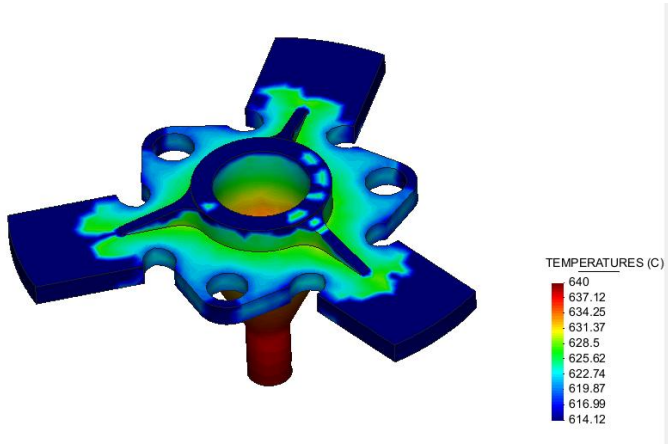


Figure H.39. Temperature evolution during filling.

Here we can see the temperatures and the temperatures colour scale, and also the time step since the start of filling.

Now we will load the second **foundry operation** we had run: solidification.

Solidification goes from the end of filling until the casting is completely solid.

Let's see the results. Go to:

Vulcan results → **Load results…** and load the **Thermal1** operation:

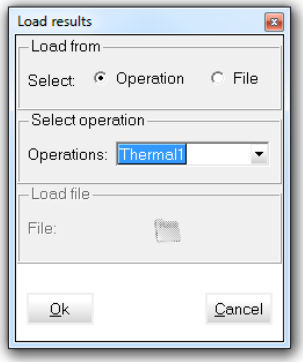


Figure H.40. Thermal operation loading.

Once the results are loaded, click Ok and let's see some solidification results.

This time we will see temperature evolution in both casting and mould, excess temperature on the cores, etc.

Go to:

Vulcan results → **Temperature evolution**

The result will be like this:

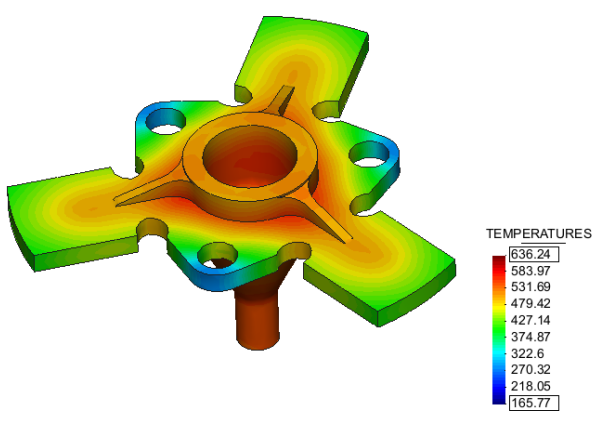


Figure H.49. Temperature evolution.

In this result, we can animate and see the temperature evolution in the casting surface, but ti would be interesting to “cut” the casting and see the temperature inside.

To do so, we will make a cut of the casting and mould.

Go to:

Vulcan results → **Results evolution with graphs**… or select the button

Select **Cut xz plane** button, and then draw a line dividing the mould horizontally in two equal halves. Selecting the Show cut sections, the result will be:

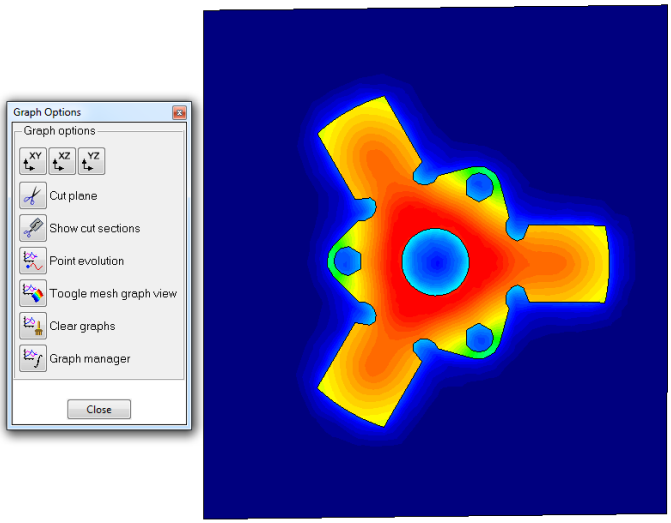


Figure H.41. Temperature in cut.

On the figure, we can see the temperature evolution of both the casting and the mould, in the **Cut xz plane**. By pressing the **Play** button on the animate window, we can see all the temperature evolution during solidification (select render normal to see a better resolution).

Now, we will add a virtual thermocouple in a point in the middle of the casting.

Go to:

View results → **Graphs** → **Point evolution** → **Temperatures** or select the button

And select a point in the middle of the casting :

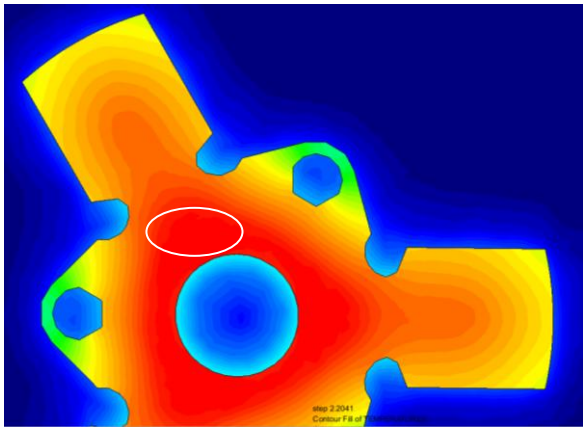


Figure H.42. Point selection.

The result will be:

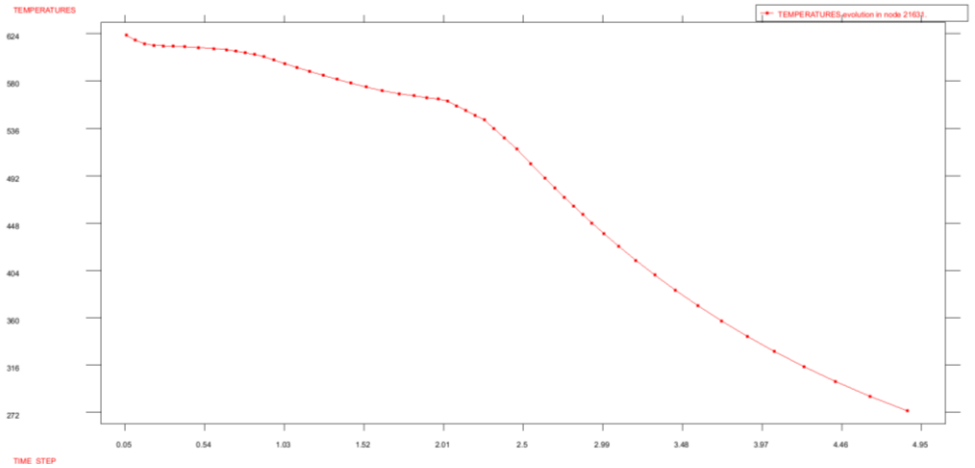


Figure H.43. Virtual thermocouple in the casting.

In the previous temperature images we cannot see the casting phase change, from liquid to solid. In order to see this and, for example, the effectivity of our riser system, the evolution of the solid phase, the last points in the casting to reach solid state, etc, we will do the following:

Go to:

Vulcan results → Solid fraction evolution or select the button

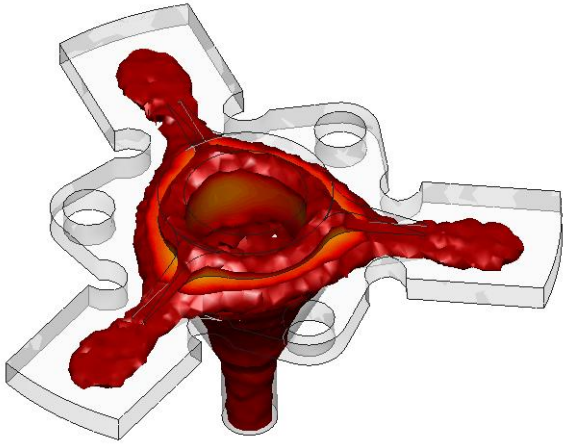


Figure H.44. Solid fraction evolution.

Here we have the liquid fraction of the casting is plotted in red, and the solid part of the casting disappears while is forming, so we only see the remaining liquid parts inside the casting as it solidifies, during the animation. The last points remaining in red should be the risers.

Finally, we will plot the **solidification modulus** of our casting. The modulus is the volume-to-surface ratio of the casting, and it can also be seen inside the casting by making cuts. Let's go back to **¡Error! No se encuentra el origen de la referencia.** and make again the temperature evolution, and then again temperature in cut. Once we have the results like in *Figure H.41*, go to:

View results → Contour fill → Solidif Modulus

The result will be:

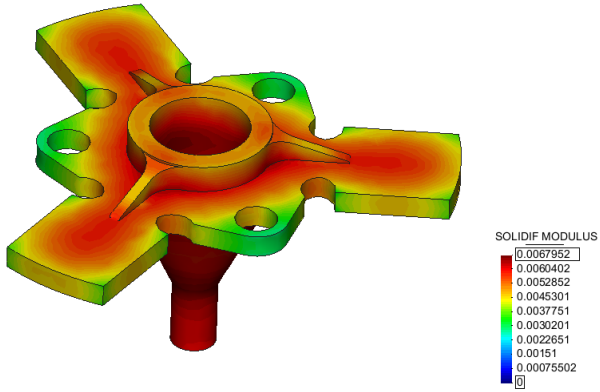


Figure H.45. Solidification modulus.